ROMI®

PROGRAMMING AND OPERATION MANUAL

ROMI DCM 620 5X

CNC SIEMENS 840D

U23672

INDÚSTRIAS ROMI S/A

MARKETING DIVISION::
Coriolano st, 710 Lapa
05047-900 São Paulo - SP - Brasil
Phone (11) 3873-3388

HEADQUARTERS:
Pérola Byington Av. , 56
13453-900 Santa Bárbara D'Oeste - SP - Brasil
Phone (19) 3455-9000

ÍNDICE

PART I - ISO PROGRAMMING

1 - PRESENTATION ____ 2
1.1 - REQUIREMENTS BEFORE PROGRAMMING... 2
2 - INTRODUCTION TO PROGRAMMING 3
2.1 - DATA BLOCKS 3
3 - FILES AND PROGRAMMING MANEGEMENT ____ 4
4 - SYSTEM OF COORDINATES ____ 5
4.1 - ABSOLUTE COORDINATES 6
4.2 - INCREMENTAL COORDINATES 6
4.3 - POLAR COORDINATES 7
5- FUNCTIONS: D, S, T, M6 / CHANGE ____ 8
6- FUNCTIONS: SLASH (/), N, MSG, POINT AND COMMA (;) ____ 9
7- PREPARATION FUNCTIONS ____ 10
8 - MISCELLANEOUS FUNCTIONS ____ 13
9 - SETTING FLOWCHART ____ 14
10 - INTERPOLATION FUNCTIONS ____ 15
10.1 - FUNCTION: G00 - FAST POSITIONING 15
10.2 - FUNCTION: G01 - LINEAR INTERPOLATION 15
10.3 - G02 AND G03 FUNCTIONS - CIRCULAR INTERPOLATION 16
10.3.1 - TURN function - Helical Interpolation 17
10.4 - FUNCTIONS: CHF/CHR AND RND/RNDM 20
10.5 - FUNCTIONS: G331 AND G332 - THREADING STEP BY STEP WITH RIGID TAP 22
11 - DWELL ____ 23
11.1 - FUNCTION: G04 23
12 - COMPENSATION FUNCTIONS ____ 24
12.1 - FUNCTIONS: G40, G41 E G42 24
13 - POLAR COORDINATE SYSTEM ____ 26
13.1 - FUNCTION: G111 26
14 - OTHERS PREPARATORY FUNCTIONS ____ 27

14.1 - FUNCTIONS: G17, G18 E G19 27
14.2 - FUNCTIONS: G500, G53 AND SUPA 27
14.3 - FUNCTIONS: G54 A G59 E G507 A G599 28
14.4 - FUNCTIONS: G09 AND G60 28
14.5 - FUNCTIONS: G64, G641, G642 28
14.6 - FUNCTION: G70 29
14.7 - FUNÇÃO: G71 29
14.8 - FUNÇÃO: G90 29
14.9 - FUNCTION: G91 30
14.10 - FUNÇÃO: G94 30
14.11 - FUNÇÃO: G95 30

15 - REPETITIONS AND JUMPS _____ 31

16 - SUBPROGRAM CALL _____ 34

17 - MCALL - SUBPROGRAM AND CYCLE MODAL CALL. _____ 36

18 - OFFN FUNCTION. _____ 38

19 - CYCLES. _____ 39
19.1 - CYCLE81 40
19.2 - CYCLE82 42
19.3 - CYCLE83 44
19.4 - CYCLE84 47
19.5 - CYCLE85 50
19.6 - CYCLE86 52
19.7 - CYCLE70 54
19.8 - HOLES1 57
19.9 - HOLES2 59
19.10 - CYCLE801 61
19.11 - LONGHOLE 63
19.12 - SLOT1 65
19.13 - SLOT2 68
19.14 - POCKET3 71
19.15 - POCKET4 74
19.16 - CYCLE76 77
19.17 - CYCLE77 80
19.18 - CYCLE61 83
19.19 - CYCLE72 85

20 - FRAME FUNCTIONS _____ 87
20.1 - FUNCTIONS: TRANS, ATRANS 87

20.2 - FUNCTION: ROT, AROT 89

21 - WORKPIECE POINTS 94

22 - ROTARY TABLE (AXIS B / AXIS C) 96
22.1 - WORK WITH 5 FACES 97
22.1.1 Examples of work using 5 FACES: 97
22.2 WORK WITH 5 AXES 99
22.2.1 Cycle 800 (Revolution Plan) 100
22.2.2 Setting example 102
22.2.3 Graphic simulation 105

23 - POWER GRAPHIC 106

24- PROGRAMMING MOLDS AND DIES: 107
24.1 - INTERACTIVE SCREEN TO CONTROL THE MOLD AND DIE FUNCTIONS 107
24.2. Examples of CYCLE 832 programming: 109

25 - “R“ CALCULATION PARAMETER 110
25.1 - “R” CALCULATION PARAMETER 110
25.1.1 - Explanation: 110
25.1.2 - Application: 110
25.1.3 - Values attribution: 110
25.1.4 - Parameter application in a program: 111
25.1.5 - Operation / arithmetic functions: 111
25.2 - OPERATORS / ARITHMETIC FUNCTIONS 112
25.2.1- Main operators and arithmetic functions: 112
25.2.2 - Arithmetic function ATAN2(___,___) 112
25.3 - OPERATORS AND LOGIC COMPARATORS 113
25.3.1 - Comparison operators: 113
25.3.2 - Logic operators: 113
25.4 - PARAMETRIC PROGRAM EXAMPLES. 114

PART II - OPERATION

1 - COMMAND PANEL - CNC SIEMENS 840 129
1.1 - DISPLAY PANEL 130
1.2 - PROGRAMMING PANEL 130
1.3 - OPERATION PANEL 131

2 - INITIAL OPERATIONS 135
2.1 - TURN MACHINE ON 135
2.2 - REFERENCE MACHINE AXES: 135
2.2.1 - Reference axis : 135
2.2.1 - Reference axis by axis : 135
2.2.2 - TURN MACHINE OFF 135

2.3 - MOVE AXES MANUALLY 136
2.3.1 - Move axes via Continuous Jog (1) 136
2.3.2 - Move axes via Continuous Jog (2 - remote panel). 136
2.3.3 - MOVE AXES VIA ELECTRONIC CRANK 137
2.3.4 - MOVE AXES VIA INCREMENTAL JOG 137
2.4 - FUNCTION T,S,M. 138
2.4.1 - Change tools. 138
2.4.2 - Turn the spindle ON. 139
2.4.3 - SPINDLE ORIENTATION. 139
2.4.4 - Miscellaneous functions 139
2.4.5 - WORK COORDINATE SYSTEM (G54 - G57 ou G505 - G599). 139
2.4.6 - Measurement unit reference 140
2.5 - RAPID TRAVERSE POSITIONING THROUGH THE “POSITION” FUNCTION 140
2.6 - LOAD THE TOOLS AT THE MAGAZINE (ATC) THROUGH THE SPINDLE 141

3 - MANUAL DATA INPUT (MDA) 142

4 - PROGRAM EDIT 143
4.1 - CREATE A NEW DIRECTORY 143
4.2 - CREATE A NEW PROGRAM 144
4.3- RENAMING A PROGRAM 144
4.4 - COPYING PROGRAM. 144
4.5 - DELETE A PROGRAM OR DIRECTORY 144
4.6 - SELECT AN EXISTING PROGRAM IN THE DIRECTORY 145
4.7 - INSERTING NUMBER OF BLOCKS AUTOMATICALLY. 145
4.8 - SEARCHING FOR DATA WITHIN A PROGRAM 145
4.9 - COPYING DATA: 146
4.10 - DELETING PROGRAM DATA 146
4.11 - DELETING PROGRAM BLOCKS: 146
4.12 - PROGRAMMING CYCLES IN INTERACTIVE PAGE. 146
4.13 - CHANGE SOME DATA IN THE INTERACTIVE PAGE. 146

5 - DATA COMMUNICATION 147
5.1 - COMMUNICATION THROUGH USB 147
5.1.1 - Recommended hardware for reading and writing USB : 147
5.1.2 - View files in the USB. 148
5.1.3 - Load a program from memory card (USB). 148
5.1.4 - Saving program from memory card. 148
5.2 - COMMUNICATION THROUGH ENTHERNET 148
5.2.1 - Recommended hardware for reading and writing ETHERNET : 149
5.2.2 - View files in the PC. 149
5.2.3 - Load the files in the machine. 149
5.2.4 - Saving program from PC. 149

6 - PROGRAM TEST 150
6.1 - EXECUTE GRAPHICAL TEST (WAY 1). 151
6.2 - EXECUTE GRAPHICAL TEST (WAY 2). 152
6.3 - TESTING PROGRAMS WITHOUT MOVING THE AXES 152
6.4- EXECUTING PROGRAM TEST IN DRY RUN MODE (RAPID FEED - DRY): 153

7 - TOOL REFERENCE 154
7.1 - CREATE A NEW TOOL 154
7.2 - DELETE TOOL. 156
7.3 - TOOL PRESET 157
7.3.1 – Tools present done in the machine 157
7.3.2 - Tool referencing made out of the machine 160
7.4 - FILL THE TOOLS DATA. 161

8 - TOOL WEAR OFFSET. 162
8.1 - TOOL WEAR COMPENSATION - “Z” AXIS. 162
8.2 - TOOL WEAR COMPENSATION - “RADIUS” 162
8.3 - PROCEDURE TO CREATE A NEW OFFSET. 163

9 - WORKPIECE REFERENCE DEFINITION 164
9.1 -WORKPIECE REFERENCE ON PART EDGE. 164
9.2 - WORKPIECE REFERENCE IN INCLINED PART. 165
9.3 - WORKPIECE REFERENCE IN A HOLES CENTER. 167
9.4 - WORKPIECE REFERENCE IN A CIRCULAR SPIGOT. 168
9.5 - WORKPIECE REFERENCE IN CENTER OF PART. 169
9.6 - WORKPIECE REFERENCE (G54 - G59 E G505 - G599) ADJUST 170

10 - PROGRAM EXECUTION 171
10.1 - RUN A PROGRAM FROM MACHINE MEMORY 171
10.2 - RUN A PROGRAM FROM USB (MODE 1). 171
10.3 - EXECUTING A PROGRAM FROM USB (MODE 2). 171
10.4 - ABORT PROGRAM EXECUTION 172
10.5 - RESTARTING IN THE MIDDLE OF PROGRAM. 172
10.5.1 - Restarting in the middle of program (Shopmill). 172
10.5.2 -Restarting in the middle of program (ISO Mode). 172
10.5.3 - Start in the midle of program using Shopmill 173
10.5.4 - Start in the middle of program interrupted during the execution. 173
10.5.5 - Start in the middle of program in the memory card 173
10.6 -INTERRUPT / RESTART PROGRAM EXECUTION. 174

11 - TOOL USEFUL LIFE MONITORING. 175
11.1 - USEFUL LIFE MONITORING BY PARTS AMOUNT. 175
11.2 - LIFETIME MONITORING IN MINUTES. 176

PART I

ISO PROGRAMMING

1. PRESENTATION

Numerical commanded machine: this is the one that has an electrical-electronic equipment, named here in as the “command”, which enables it to run an automatic sequence of activities.

To perform part machining via CNC machine tool, two items must be used as reference:

a) A program must be prepared based on a part drawing, by using commands interpreted by the CNC. These commands are described in this manual, Part 1 - Programming.

b) The program must be read by the CNC. Tools must be prepared for the part according to the programming prepared, and then the machining process must be performed. These processes are described in this manual, Part 2 - Operation.

1.1 - REQUIREMENTS BEFORE PROGRAMMING...

A - Part Drawing Review: Blank and Finished

Perform an analysis on the feasibility of the part execution, considering the dimensions required, amount of material to be removed, required tools, material fastening, etc.

B - Methods and Processes Review

Define the machining phases of the part to be executed, thus establishing what to do and when to do.

C - Tool Selection

The selection of good tools is crucial for the good utilization of the equipment, as well as its position on the magazine so to minimize change times.

D - Be Familiar with the Physical Parameters of the Machine and its Programming

It is necessary to know all the programmong resources available and the chip removal capability, as well as maximum speed and number of tools, aiming at minimizing programming and operation times.

E - Definition of Cut Parameters

By considering the material to be machined, search for the cut data with the tool manufacturer:

Feeds, speed and cut depth.

2 - INTRODUCTION TO PROGRAMMING

This manual has been prepared only to cover the basic command functions, aiming to simplify programming and operation.

We inform here in that, as this is a modular command, some functions presented here may not be included in the machine configuration.

2.1 - DATA BLOCKS

These are clusters of command and positioning functions in a single record, in order to run, step by step, the sequential program order..

A block is comprised by a sequential (optional) number, command functions and EOB code in its end, and it is displayed on video as ;

The block has the configuration below

N_____ G_____ X _____ Y_____;
N_____ T_____ ;
N_____ M_____ ;

Where:

Function N_____	= Sequential number
Function G_____	= Preparation function
Functions X_____ Y_____	= Positioning functions
Function T_____	= Selects tool
Function M_____	= Miscellaneous functions

The complete syntax of each function will be described later.

3 - FILES AND PROGRAMMING MANEGEMENT

To be more fexible with data (programs and folders), they can be found, stored and organized according different criterion.

The files and programs are stored in diferents directories, they will be stored in folders according to their functions and characteristics.

Directories examples:

- Programs
- Subprograms
- User cycles
- Siemens cycles

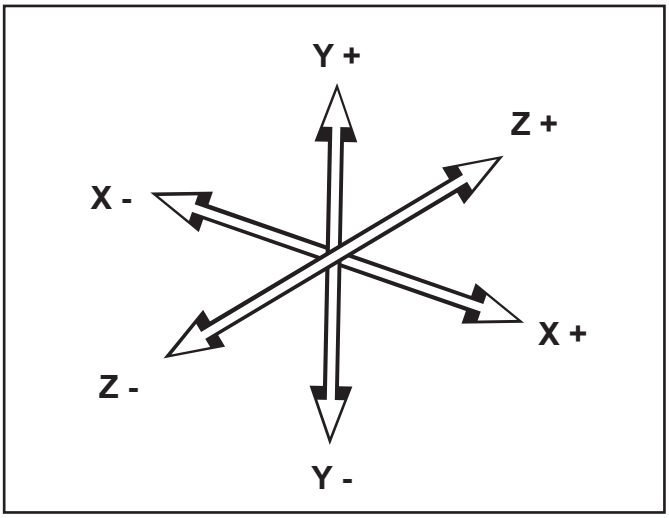
Each program corresponds to a file and each file has an extension which will inform the file type used.

Extension examples :

- CMA Machine manufacturer cycles
- CST Siemens cycles
- CUS User cycles
- MPF Main programs
- SPF Subprograms
- INI Start up files (tool data)

4 - SYSTEM OF COORDINATES

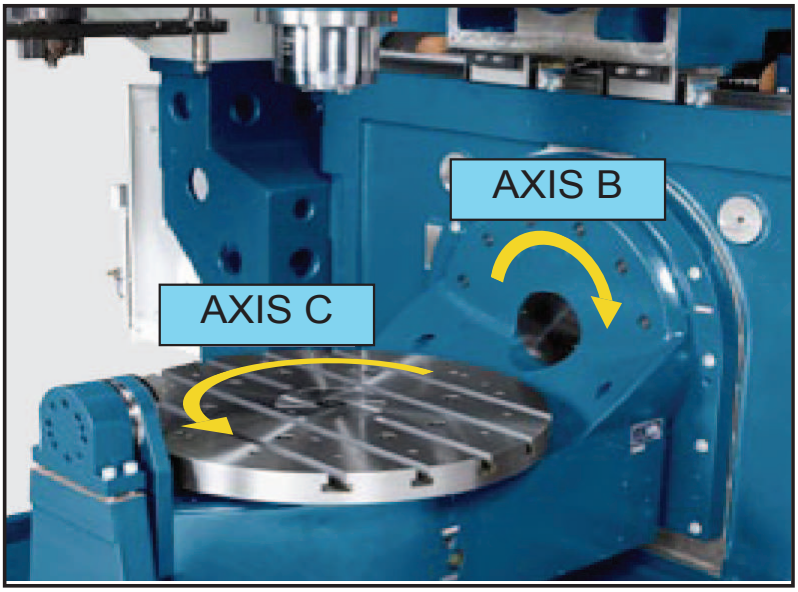
In order to the machine become able to work with the specified positions, these shall be declared in a reference system, which corresponds to the displacing directions of the carriage (axes X, Y, Z). We use, for this purpose, the Cartesian coordinate system:



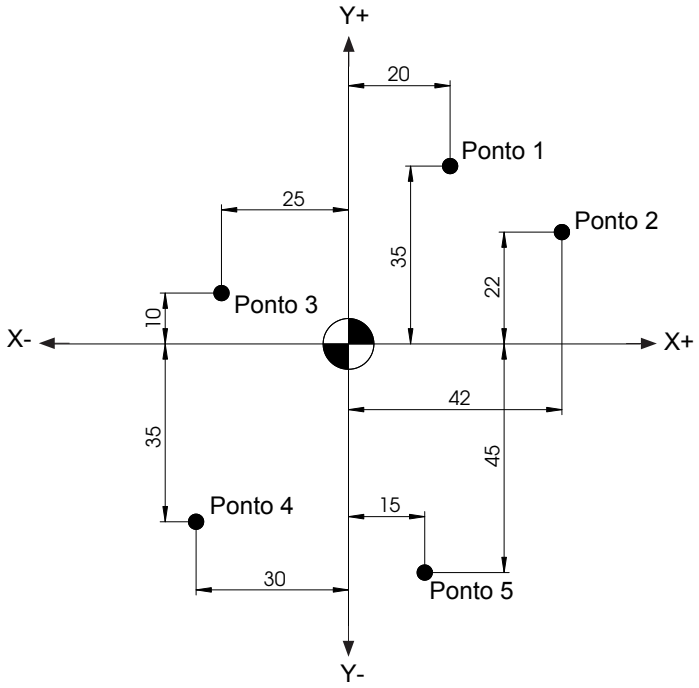
The position of the coordinate system in relation to the machine depends on the type of machine.

Machines from DCM line have a “MOBILE COLUMN” system, that is, the upward displacement corresponds to the movement of the own tool, maintaining the saddle of the machine static.

This version of equipment also has setting axes known as AXIS B and AXIS C, which perform the rotary movement of the saddle. These axes shall be set in degrees.



4.1 - ABSOLUTE COORDINATES

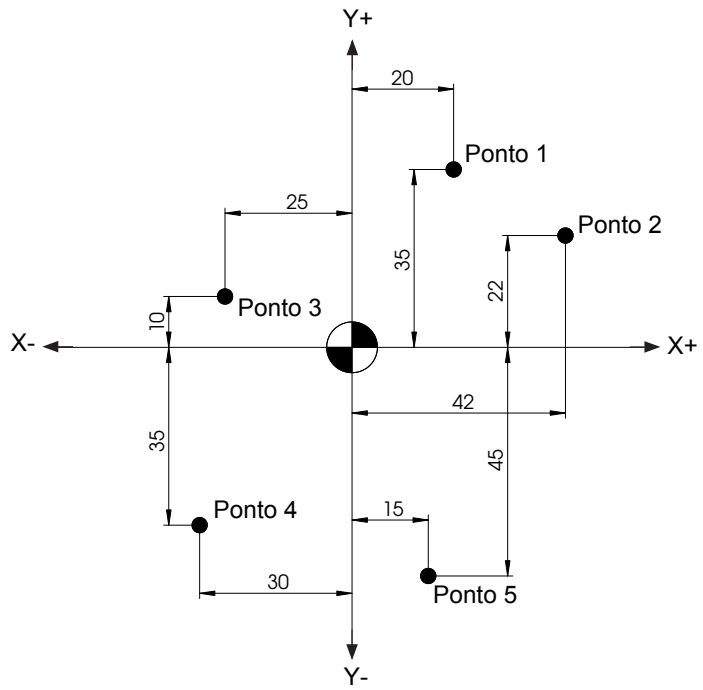


In the absolute coordinate system, the axes' positions are measured from the predefined zero-part, and we must always inform the position to where the tool must be moved to program in this system.

Example:

Ponto	Eixo X	Eixo Y
1	20	35
2	42	22
3	-25	10
4	-30	-35
5	15	-45

4.2 - INCREMENTAL COORDINATES

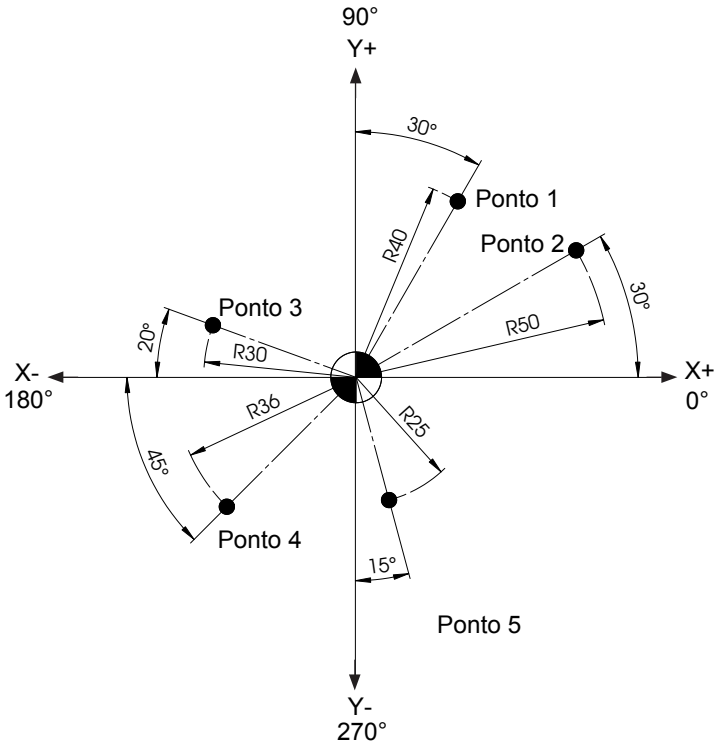


In the incremental coordinate system, axes' positions are measured from the position previously established, and we must inform the distance to be traveled by the tool as from the current position in order to program in this system.

Example:

Ponto	Eixo X	Eixo Y
1	20	35
2	22	-13
3	-67	-12
4	-5	-45
5	45	-10

4.3 - POLAR COORDINATES



So far, the method to determine points was described based on a Cartesian coordinate system, but there is another way to declare the points: in function of angles and radius.

This programming mode is named as polar coordinate system.

Example:

Point	Radius	Degree
1	40	60
2	50	30
3	30	160
4	36	225
5	25	285
Pólo	X0	Y0

5- FUNCTIONS: D, S, T, M6 / CHANGE

Explanation:Tool number select, tool geometry spindle orientation.

The addres “T” is used to select and change the tool possition. In the “D” line (CNC Siemens 828), it’s possible store 30 different tools.

To change the tool, it’s necessary programming the “M6” code together the “T” function.

For each tool can be created until 9 tool correctors. These correctors are activated with D code. For example: To active the corrector number 2, the code D2 should be programmed.

To active the spindle revolution (RPM), the S code should be programmed together the value of desired revolution.

Example:

T01	(CALL THE TOOL NUMBER 1)
M6	(EXECUTE THE CHANGE)
D01	(ACTIVE THE LENGTH CORRECTOR NUMBER 1)
S1500 M3	(ACTIVE THE SPINDLE REVOLUTION WITH 1500 RPM)

6- FUNCTIONS: SLASH (/), N, MSG, POINT AND COMMA (;)

Explanation: Inhibit blocks execution, sequencial blocks numbers, messages to the operator and help comments.

We use the slash **" / "** function when it's necessary to inhibit blocks execution in the program, without change the program.

If the slash**" /"** is put in the begin of some blocks, these blocks won't be executed for the control, since the operator selects **"SKP: SKYP BLOCK"**function. If this option isn't selected, the blocks will be executed.

Examples:

N10_____ (executed block)
/N20_____ (don't executed block)
N30_____ (executed block)
/N40_____ (don't executed block)
/N50_____ (don't executed block)
N60_____ (executed block)

The purpose of the "N" function is to make the sequencial numeration of program blocks. You can use this or not, it means, the "N" programming is optional.

Examples:

N10 ____
N20 ____
N30 ____

During the program edition can be necessary to insert comments to help the operator. These comments will be ignored for the control.

In the comments beginning should be put point and comma **" ; "**

Examples:

;TEST_PART
N30 T02**;** END MILL
N100 M30**;** END OF PROGRAM

During the program execution can be programmed messages to inform the operator what is happening in the program or to inform any important thing.

The messages are programmed using the MSG("desired message") function, and the characterses limit are 124.

To cancel a menssage should be programmed the **MSG ("")** function.

Examples:

N10 **MSG ("EXTERNAL CONTOUR ROUGHING")**
:
N100 **MSG ("")**

7- PREPARATION FUNCTIONS

Preparation functions are those which define to the machine which type of displacement it will perform.

They are split in two groups:

a) Modal - The function stays active, until another modal function of the same group is set.

b) Non-modal - The function stays active in the block where it was set only.

Example:

N100 G01 X100 F1000
N110 Y30
N120 X40
N130 G00 Z15

Function G01 stays active from block N100 to block N120.

In block N130, it is cancelled by function G00, because both are modal, belonging to the same group.

Below there is a table with the main Preparation Functions applicable to set machines from “DCM” Line with CNC Siemens 840D.

PREPARATION FUNCTIONS				
FUNCTION	DESCRIPTION	GROUP	MODAL	
			Y	N
G00	Quick Positioning	01	X	
G01*	Linear Interpolation	01	X	
G02	Circular Interpolation – clockwise direction	01	X	
G03	Circular Interpolation – counterclockwise direction	01	X	
G331	Tapping with rigid tapping	01	X	
G332	Regress movement of rigid tapping	01	X	
G04	Dwell time	02		X
G111	Setting in polar coordinates	03	X	
G17*	Selects working plan XY	06	X	
G18	Selects working plan XZ	06	X	
G19	Selects working plan YZ	06	X	
G40*	Cancels tool radius offset	07	X	
G41	Activates tool radius offset at left of the profile	07	X	
G42	Activates tool radius offset at right of the profile	07	X	
G500	Deactivates the work coordinate system (G54…G599), activating the coordinates system of the machine	08	X	
G53 / SUPA	Machine Coordinate System	08		X
G54	1st Work Coordinate System	08	X	
G55	2nd Work Coordinate System	08	X	

PREPARATION FUNCTIONS				
FUNCTION	DESCRIPTION	GROUP	MODAL	
			Y	N
G53 / SUPA	Suspends the work coordinate system for the current block	08		X
G54	Activates the 1st work coordinate system	08	X	
G55	Activates the 2nd work coordinate system	08	X	
G56	Activates the 3rd work coordinate system	08	X	
G57 G58 G59	Activates the 4th work coordinate system Activates the 5th work coordinate system Activates the 6th work coordinate system	08	X	
G507	Activates the 7th work coordinate system	08	X	
G5xx	Activates the “xx°” work coordinate system (from G5050 to G599)	08	X	
G599	Activates the 99th work coordinate system	08	X	
G09	Activates exact stop for the current block	10		X
G60*	Activates exact stop mode, leaving live edges and canceling the contour mode	10	X	
G64	Activates the contour mode with "look-ahead", chamfering edges	10	X	
G641	Activates the contour mode with "look-ahead", rounding edges	10	X	
G642	Activates the contour mode with "look-ahead", rounding edges in spline shape	10	X	
G70	Activates English measuring system (inches)	13	X	
G71*	Activates metric measuring system (millimeter)	13	X	
G90*	Activates absolute coordinate system	14	X	
G91	Activates incremental coordinate system,	14	X	
G94*	Infeed set in millimeter/minute or inches/minute	15	X	
G95	Infeed set in millimeter/revolution or inches/revolution	15	X	
CFC*	Activates infeed control by the tangency point of the tool, with parts in internal and external edges	16	X	
CFTCP	Activates infeed control by the tool center	16	X	
CFIN	Activates infeed control by the tool tangency point with the part for internal edges and by tool center for external edges	16	X	
BRISK	Ordinary mode of axes acceleration	21	X	
SOFT	Axes acceleration soft mode	21	X	
FFWOF	Activates control mode of the next infeeds	24	X	
FFWON	Deactivates control mode of next infeeds	24	X	
COMPON	Activates the block compressor system	30	X	
COMPCAD	Activates block compressor system	30	X	

PREPARATION FUNCTIONS				
FUNCTION	DESCRIPTION	GROUP	MODAL	
			Y	N
COMPCURV	Activates block compressor system for program using polynomial equations	30	X	
COMPOF	Deactivates block compressor system	30	X	

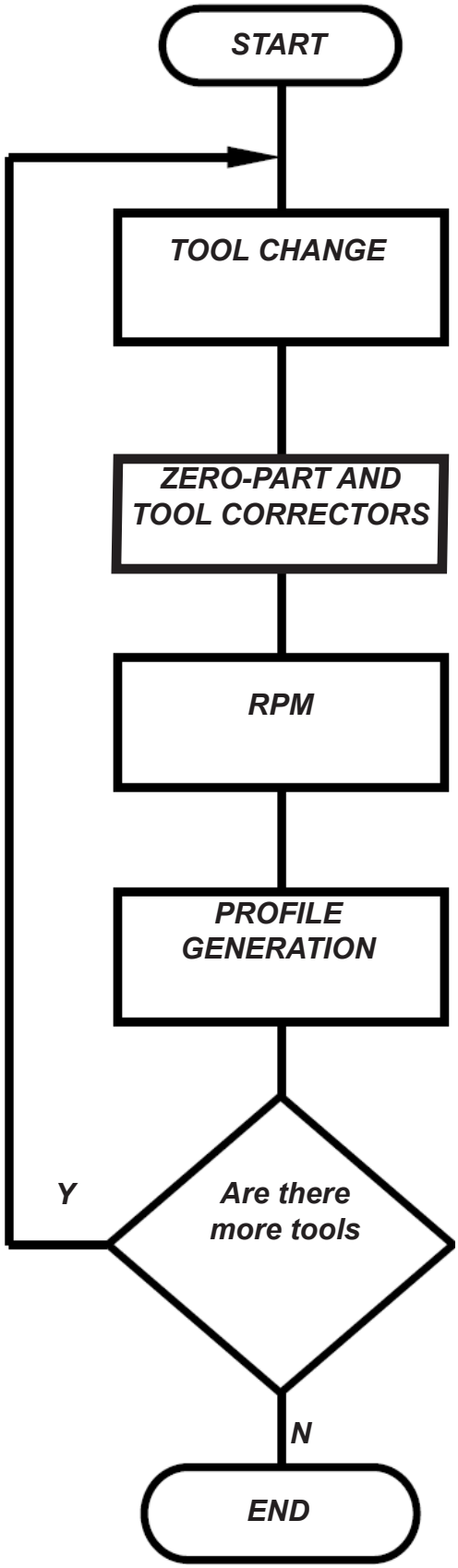
8 - MISCELLANEOUS FUNCTIONS

Miscellaneous functions, also known as code M, are functions which make interface between user and machine hardware, by the interaction with the ladder developed by its manufacturer.

Below there is a table with the main miscellaneous functions

FUNÇÕES MISCELÂNEAS	
FUNÇÃO	**DESCRIÇÃO**
M00	Mandatory stop of program
M01	Optional stop of program
M02	End of program
M03	Turns on the spindle in clockwise direction
M04	Turns on the spindle in counterclockwise direction
M05	Turns off the spindle
M06	Perform tool change
M07	Turns on cut coolant by spindle center (optional)
M08	Turns on cut coolant by hoses external to spindle
M09	Turns off cut coolant (M07 and M08)
M17	End of subprogram
M30	End of program
M36	Opens automatic door (optional)
M37	Closes automatic door (optional)
M45	Turns on guards cleaning system (optional)
M46	Turns off guards cleaning system (optional)
M47	Turns on chip conveyor (optional)
M48	Turns off chip conveyor (optional)
M50	Unlocks axis C brake
M51	Locks axis C brake
M52	Turns on parts measuring and inspection system (optional)
M53	Turns off parts measuring and inspection system (optional)
M54	Turns on air cooling system (optional)
M55	Turns off air cooling system (optional)
M58	Unlocks axis B brake
M59	Locks axis B brake

9 - SETTING FLOWCHART



• START

G17 G71 G90 G94 ; - *Safety block*

• TOOL CHANGE

G53 G0 Z0 D0 ; - *regress the spindle near to the point and changes tool*
T_ _ ; - *selects tool*
M06 ; - *changes tool*
G64 CFIN ; *activates look ahead and infeed control*

• ZERO-PART AND TOOL CORRECTORS

G5_ D_ ; - *Activates zero-part (G54-G59 and G505 to G599) and activates tool correctors (D1 to D9).*

• RPM SETTING

S_ _ _ _ M_ _ ; - *Determines the RPM value (S___) and turns on the spindle (M03 or M04)*

• PROFILE GENERATION

G0 B_ _ C_ _ ; - *Saddle/table positioning, in degrees*
G0 X_ _ Y_ _ ; - *Approach on X and Y axes*
Z_ _ _ ; - *Approach on Z axis*
:
(other instructions according the program need
:

• END OF PROGRAM

G53 G0 Z0 D0 ; - *regress spindle on Z*
G53 G0 B0 C0*; - positions saddle/table in degrees*
M30 ; - *end of program*

.

10 - INTERPOLATION FUNCTIONS

10.1 - FUNCTION: G00 - FAST POSITIONING

Explanation:

The axes move in fast feed to a given position referenced to zero program, or to an incremental distance from the current one, according to the function G90 or G91 previously established.

In “DCM” Line machines, the displacement speed in fast feed on X-, Y- and Z-axes is 40 meters per minute for all the models.

Syntax:

G00 X_____ Y_____ Z_____B_____C_____

Where:

X = Coordinate of the final point of the motion for linear X-axis
Y = Coordinate of the final point of the motion for linear Y-axis
Z = Coordinate of the final point of the motion for linear Z-axis
B = Coordinate of the final point of the motion for angular B-axis
C = Coordinate of the final point of the motion for angular C-axis
(feed in degrees/min)

10.2 - FUNCTION: G01 - LINEAR INTERPOLATION

Explanation:

The axes move in programmed feed, specified by F, to a given position referrenced to zero program, or to an incremental distance from the current one, according to the function G90 or G91 previously established. The maximum programmable feed speed is 15,000 millimeters per minuto, that is, 15 meters per minute.

Syntax:

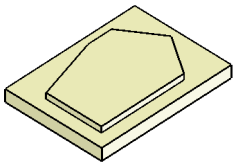
G01 X____ Y_____ Z_____ B_____C_____F_____

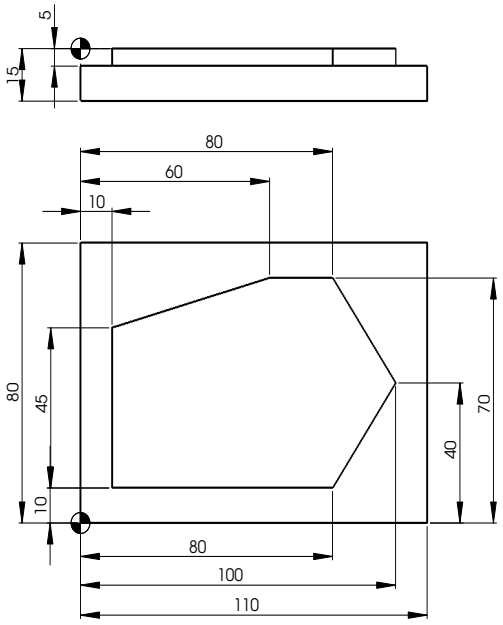
Where:

X = Coordinate of the final point of the motion for linear X-axis
Y = Coordinate of the final point of the motion for linear Y-axis
Z = Coordinate of the final point of the motion for linear Z-axis
B - Coordinate of the final point of the motion for angular B-axis
C - Coordinate of the final point of the motion for angular C-axis
(feed in degrees/min)
F = Feed speed (mm/min or mm/rotation)

Example:

```
  :                          :
G01 X10 Y10 F700           G01 X10 Y10 F700
G01 X80 Y10                X80
G01 X100 Y40               X100 Y40
G01 X80 Y70          or    X80 Y70
G01 X60 Y70                X60
G01 X10 Y55                X10 Y55
G01 X10 Y10                Y10
  :                          :
```





10.3 - G02 AND G03 FUNCTIONS - CIRCULAR INTERPOLATION

Explanation:

By using circular interpolation, arcs are generated in clockwise (G02) or counterclockwise (G03) direction.

It is necessary to define the work plan of axes for the arc.

By definition, the clockwise or counterclockwise directions have the view from the positive to negative direction of the axis that is not included in work plan.

The syntax below aimed at G02 is also valid for G03.

Syntax

a) Arc on XY plan (G17)

G02/G03 X___ Y___ CR=___ F___*
or G02/G03 X___ Y___ I___ J___ F___*
or G02/G03 X___ Y___ I=AC(___) J=AC(___) F___*

b) Arc on XZ plan (G18)

G02/G03 X___ Z___ CR=___ F___*
or G02/G03 X___ Z___ I___ K___ F___*
or G02/G03 X___ Z___ I=AC(___) K=AC(___) F___*

c) Arc on YZ plan (G19)

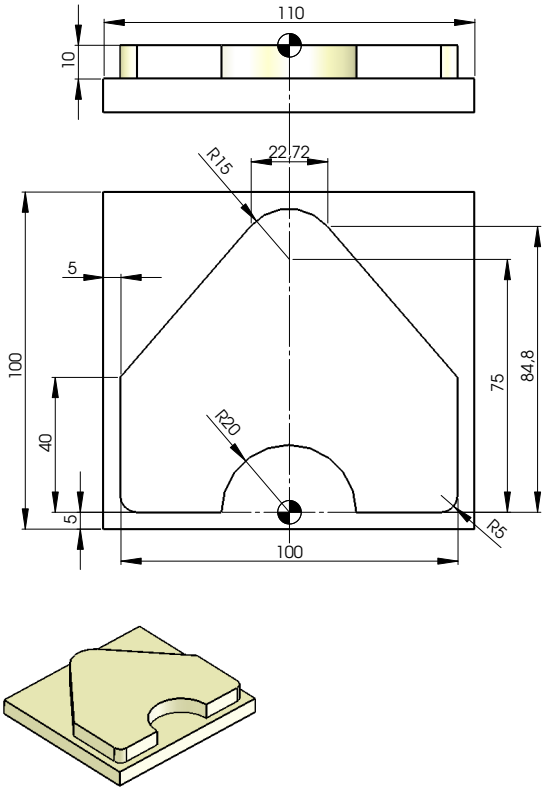
- G02/G03 Y___ Z___ CR=___ F___*
or G02/G03 Y___ Z___ J___ K___ F___*
or G02/G03 Y___ Z___ J=AC(___) K=AC(___) F___*

Where:

X	-	Final arc position in X
Y	-	Final arc position in Y
Z	-	Final arc position in Z
CR=	-	Arc radius (negative for arc gretare than 180 degrees)
I	-	Distance on X with sign (+ -) from the start point to arc center
J	-	Distance on Y with sign (+ -) from the start point to arc center
K	-	Distance on Z with sign (+ -) from the start point to arc center
I=AC(...)	-	Absolute coordinates from the arc center position in X
J=AC(...)	-	Absolute coordinates from the arc center position in Y
K=AC(...)	-	Absolute coordinates from the arc center position in Z
F	-	Feed speed along arc

Example:

```
  :
G01 X-45 Y0 F300
G02 X-50 Y5 CR=5
; or G02 X-50 Y5 I0 J5
; or G02 X-50 Y5 I=AC(-45) J=AC(5)
G01 Y40
X-11.36 Y84.8
G2 X11.36 Y84.8 CR=15
; or G02 X11.36 Y84.8 I11.36 J-9.8
; or G02 X11.36 Y84.8 I=AC(0) J=AC(75)
G01 X50 Y40
Y5
G02 X45 Y0 CR=5
; or G02 X45 Y0 I-5 J0
; or G02 X45 Y0 I=AC(45) J=AC(5)
G01 X20
G03 X-20 Y0 CR=20
; or G02 X-20 Y0 I-20 J0
; or G02 X-20 Y0 I=AC(0) J=AC(0)
G01 X-45
  :
```



NOTES:

- When XYZ coordinates are omitted and the center is specified with I, J, or K, it's considered that the end point is the same of the start point, generating a 360 degrees arc.

- When an arc is bigger than 180 degrees, the radius value should be specified with a negative value.

10.3.1 - TURN function - Helical Interpolation

The TURN function is a resource used together with circular interpolation functions (G02/G03) to generate moves in a spiral way, it means, to synchronize a circular move in a specified work plan with a third axes linear move, generating a helix.

Syntax:

Sinchronized with XY arc (G17)

G2/G3 X___ Y___ I___ J___ Z___ TURN=___ F___*
or G2/G3 X___ Y___ I=AC(___) J=AC(___) Z___ TURN=___ F___*

Sinchronized with XZ arc (G18)

G2/G3 X___ Z___ I___ K___ Y___ TURN=___ F___*
or G2/G3 X___ Z___ I=AC(___) K=AC(___) Y___ TURN=___ F___*

Sinchronized with YZ arc (G19)

G2/G3 Y___ Z___ J___ K___ X___ TURN=___ F___*
G2/G3 Y___ Z___ J=AC(___) K=AC(___) X___ TURN=___ F___*

where:

X	-	Final arc position in X
Y	-	Final arc position in Y
Z	-	Final arc position in Z
I	-	Distance on X with sign (+ -) from the start point to arc center
J	-	Distance on Y with sign (+ -) from the start point to arc center
K	-	Distance on Z with sign (+ -) from the start point to arc center
I=AC(...)	-	Absolute coordinates from the arc center position in X
J=AC(...)	-	Absolute coordinates from the arc center position in Y
K=AC(...)	-	Absolute coordinates from the arc center position in Z
TURN=	-	Total Turn Number
F	-	Feed speed along arc

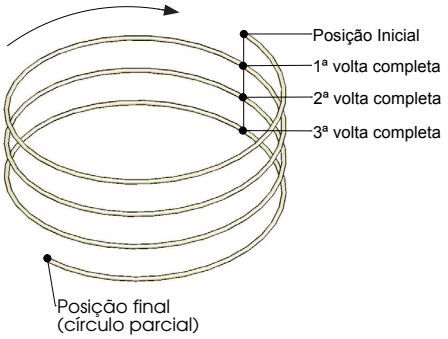
***NOTES:** Tool radius compensation is applied only for circular motion.*

Sample: G17 work plan, axes for the circular interpolation are X and Y.

The vertical linear move in the example below will be executed for **Z AXIS.**

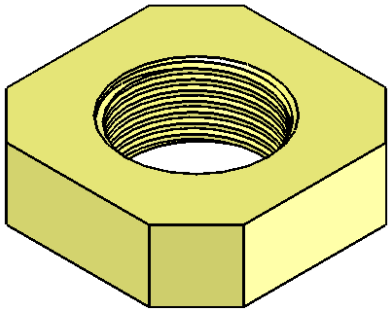
Moves sequence:

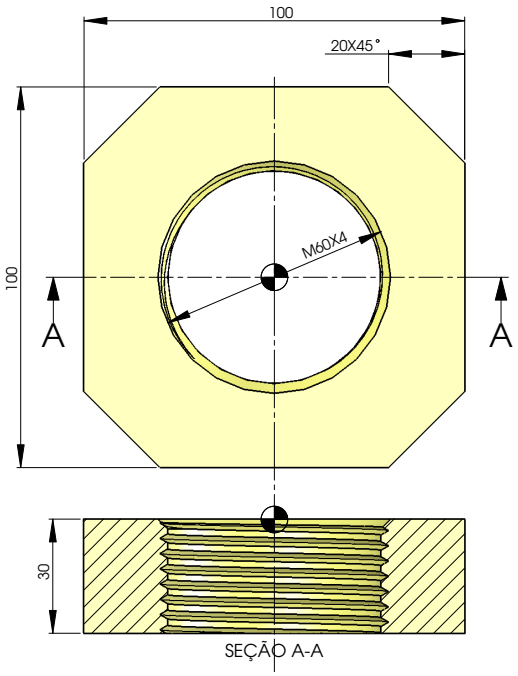
1. To position in the start point, descounting the tool radius (initial coordinate)
2. With TURN= executes the full circle programmed.
3. If necessary, goes to the circle end point through of a partial rotation.



Example:

```
  :
G0 X0 Y0
Z2
X20
G1 Z0 F350
G2 X20 Y0 Z-32 I=AC(0) J=AC(0) TURN=8
G0 X0 Y0
Z10
  :
```





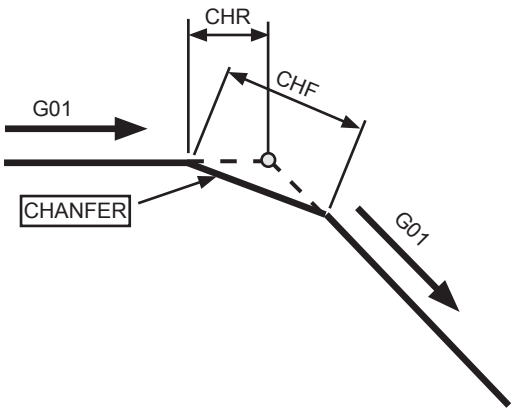
***NOTES** : In the example it was considered a 20 mm tool diameter.*

10.4 - FUNCTIONS: CHF/CHR and RND/RNDM

Explanation: Chanfer, and rounding corners.

To use these functions, program them in the same block of linear or circular interpolation in order to create, in function of the next motion, a chanfer or rounding.

The value programmed just after CHR function indicates the bevel dimension in relation to motion intersection (vertex).



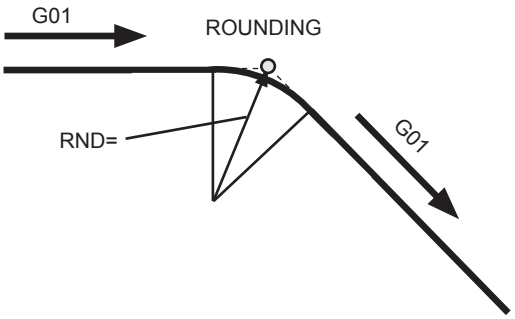
Syntax:

CHF=(...)
or
CHR=(...)

Example:

G1 X50 Y30 F100 CHR=5
G1 X100 Y20

The value programmed just after RND function indicates the corner radius..

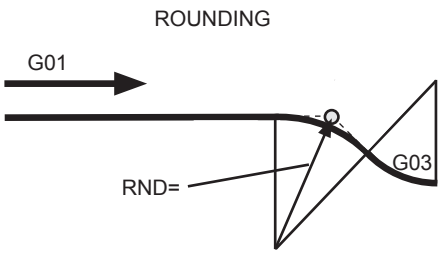


Syntax:

RND=(...)

Example:

G1 X50 Y30 F100 RND=10
G1 X100 Y20



Example:

G1 X(...) Y(...) F(...) RND=(...)
G3 X(...) Y(...) I(...) J(...)

To work with a modal rounding , it means, to allow insert after each block move, a rounding between linear contours and circular contours, we use the RNDM function.

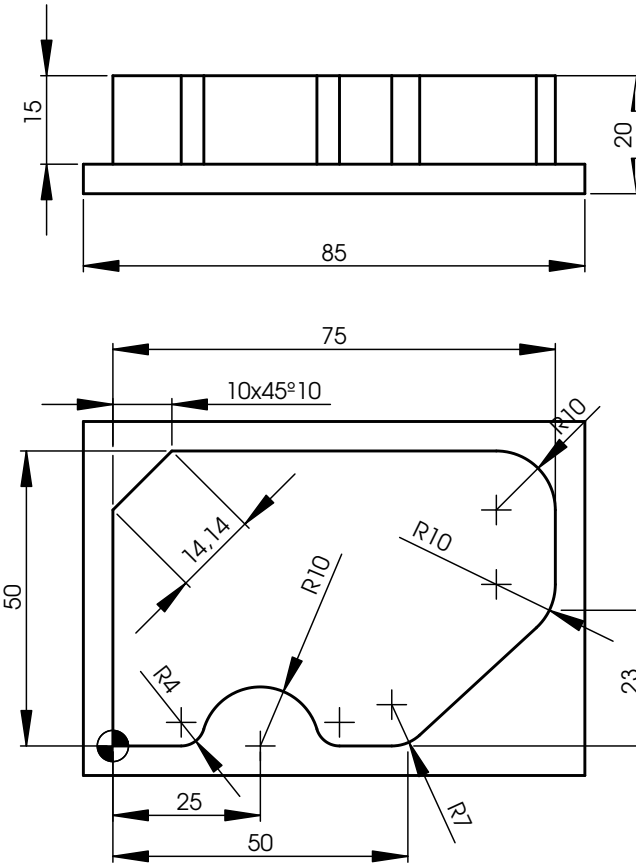
Syntax:

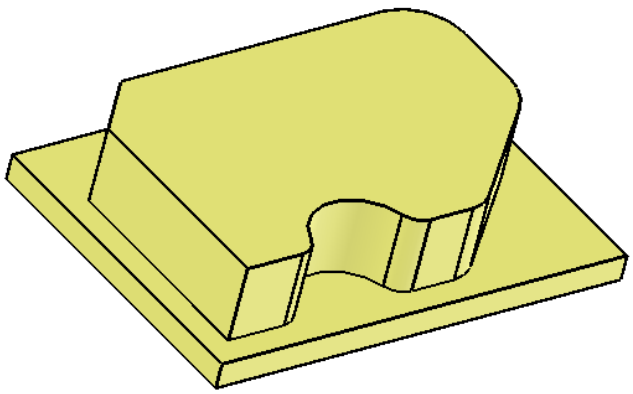
RNDM=(...)

To desactive modal rounding function should be programmed the RNDM=0 function.

Example:

```
G17 G71 G90 G94
G53 G0 Z-110 D0
T02; FRESA D16 MM
M6
G54 D1 G64 CFIN
S2000 M3
G0 X-15 Y-15
Z-15
G41 G01 X0 Y0 F700
G01 X0 Y50 CHR=10
; or G01 X0 Y50 CHF=14.14
G01 X75 Y50 RND=10
G01 X75 Y23 RND=10
G01 X50 Y0 RND=7
G01 X35 Y0 RND=4
G03 X15 Y0 CR=10 RND=4
G01 X0 Y0
G40 G01 X-15 Y-15
G53 G0 Z-110 D0
M30
```





10.5 - FUNCTIONS: G331 AND G332 - THREADING STEP BY STEP WITH RIGID TAP

Explanation:

This functions are used to make threading step by step with rigid tap without be necessary use float system.

Syntaxes:

G331 X___ Y___ Z___ I___ J___ K___
G332 X___ Y___ Z___ I___ J___ K___

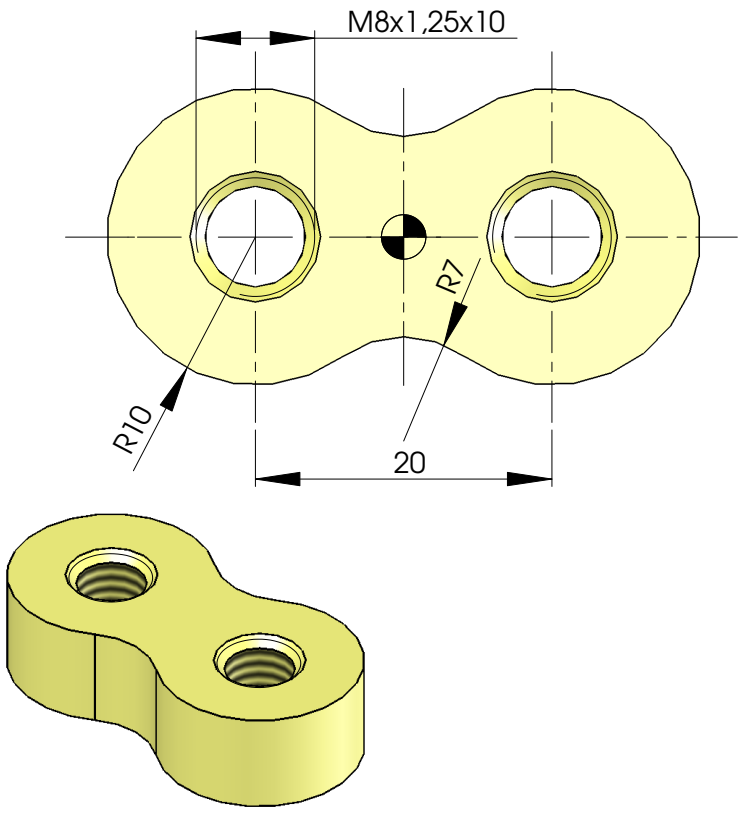
where:

- X - End point coordinate to the X axis move.
- Y - End point coordinate to the Y axis move.
- Z - End point coordinate to the Z axis move.
- I - Thread lead in the X axis direction (positive to right thread and negative to left thread)
- J - Thread lead in the Y axis direction (positive to right thread and negative to left thread)
- K - Thread lead in the Z axis direction (positive to right thread and negative to left thread)

Example: Right Thread

Thread Lead = 1.25 mm
Revolution = 800 RPM

```
  :
T15; M8X1.25
M06
G54 D1 G64 CFIN
S800 M03
G0 X-10 Y0
Z5
G95
G331 Z-10 K1.25
G332 Z5 K1.25
G0 X10
G331 Z-10 K1.25
G332 Z5 K1.25
G53 G0 Z-110 D0
  :
```



11 - DWELL

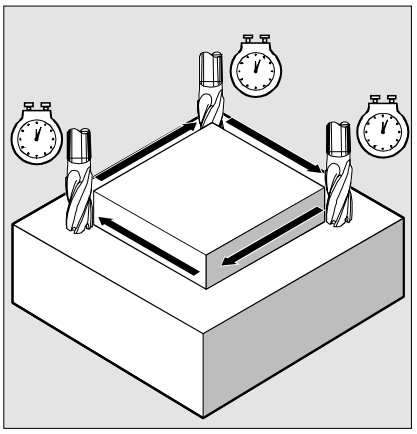
11.1 - FUNCTION: G04

Explanation: DWELL

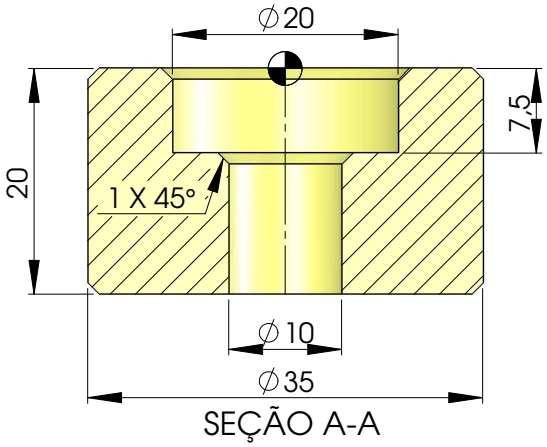
Stop the roughing between two blocks, during a specified time.

Syntax:

G4 F___ Time (seconds)
G4 S___ Numer of RPM

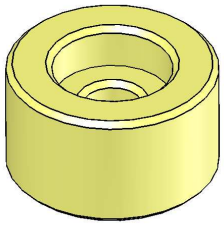


Example: Pocket step by step



SEÇÃO A-A

:
T12; END MILL Ø20 X 7.5
M06
G54 D1 G64 CFIN
S2000 M03
G00 X0 Y0
Z5
G1 Z-7.5 F440
G4 S1
;or G4 F0.03 (60 sec / 2000 rev = 0,03 sec/rev)
G1 Z5
G53 G0 Z-110 D0
:



12 - COMPENSATION FUNCTIONS

12.1 - FUNCTIONS: G40, G41 E G42

Explanation: Tool radius compensation

The tool radius compensation functions have been developed to facilitate the programming of certain contours. By using them, it is possible programs according to the drawing dimensions, without concerning on the tool radius, since these functions are in charge of calculating the tool runs, from its radius, which must be inserted in “TOOL OFFSET” page.

G40, G41 and G42 functions are used to work with radius compensation, namely:

G40 **-** Cancels tool radius compensation
G41 - Compensates tool at left of the material to be machined
G42 - Compensates tool at right of the material to be machined

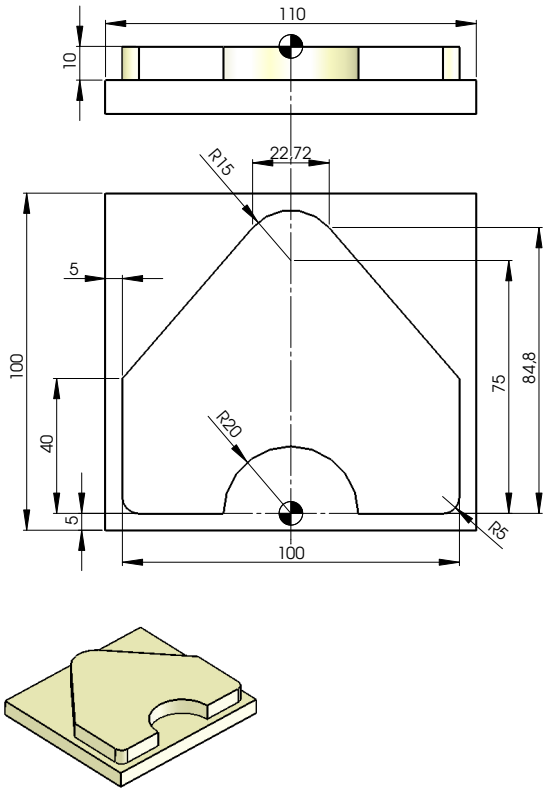
With the compensation function activated, the control automatically calculates the respectives halfway courses of the tool.

To the tool courses calculation the control needs of the following information: T (tool number) and D (corrector number).

To active or desactive the tool radius compensation G40, G41 or G42 should be programmed a position code with G0 or G1, with at least a axis move (better both).

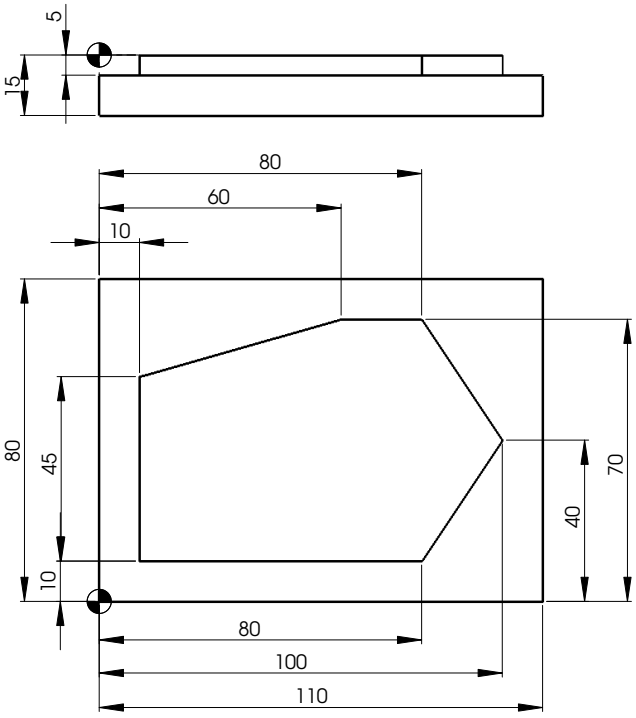
Example 1: Program with tool left compensation (G41)

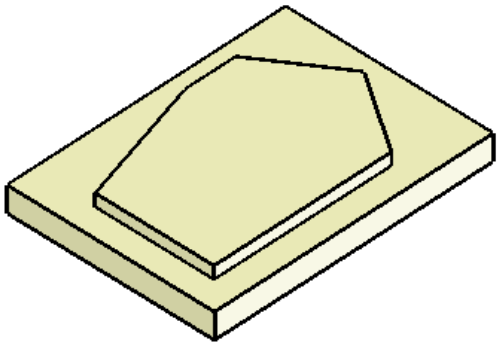
```
G17 G71 G90 G94
G53 G0 Z-110 D0
T03; MILLING D25 MM
M6
G54 D1 G64 CFIN S2500 M3
G0 X-40 Y-30
Z-10
G41 G01 X-45 Y0 F300
G02 X-50 Y5 CR=5
G01 Y40
X-11.36 Y84.8
G2 X11.36 Y84.8 I=AC(0) J=AC(75)
G01 X50 Y40
Y5
G02 X45 Y0 CR=5
G01 X20
G03 X-20 Y0 I-20 J0
G01 X-45
G40 X-40 Y-30
G53 G0 Z-110 D0
M30
```



Example 2: Program with tool right compensation (G42)

```
G17 G71 G90 G94
G53 G0 Z-110 D0
T02; D16 MM
M6
G54 D1 G64 CFIN
S2000 M3
G0 X-20 Y-20
Z-5
G42 G01 X10 Y10 F700
G01 X80 Y10
G01 X100 Y40
G01 X80 Y70
G01 X60 Y70
G01 X10 Y55
G01 X10 Y10
G40 X-20 Y-20
G53 G0 Z-110 D0
M30
```





13 - POLAR COORDINATE SYSTEM

13.1 - FUNCTION: G111

Explanation: Define the polar coordinate

The polar coordinate system is a programming mode where the coordinates are indicated via angles and radiuses.

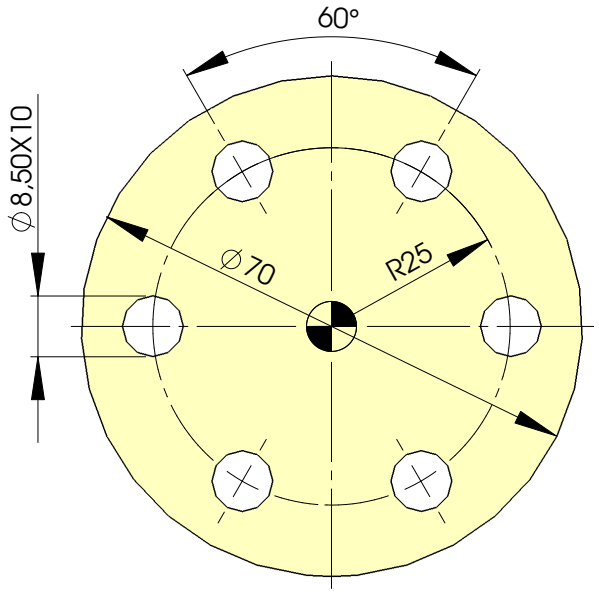
Syntax:

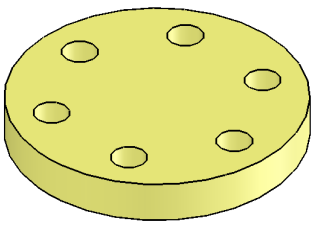
G111 X___ Y___;
G0/G1 ***AP=(...) RP=(...***)
G2/G3 ***AP=(...) RP=(...)***

where:
X - Coordinate of the polar point on X axis
Y - Coordinate of the polar point on Y axis
AP - Polar angle (horizontal angle reference)
RP - Polar radius (inch / milimeter)

Example:

```
:
G0 X0 Y0
Z10
G111 X0 Y0
G0 AP=0 RP=25
G1 Z-10 F300
G0 Z2
AP=60 RP=25
G1 Z-10
G0 Z2
AP=120 RP=25
G1 Z-10
G0 Z2
AP=180 RP=25
G1 Z-10
G0 Z2
AP=240 RP=25
G1 Z-10
G0 Z2
AP=300 RP=25
G1 Z-10
G0 Z10
:
```



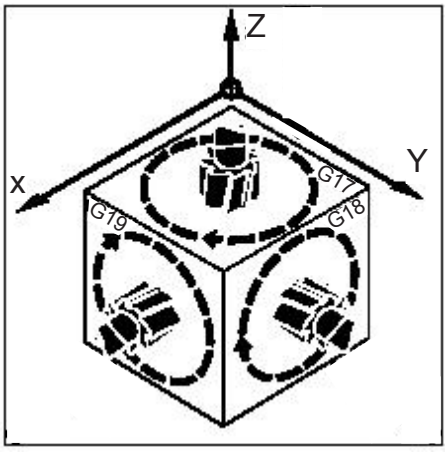


14 - OTHERS PREPARATORY FUNCTIONS

14.1 - FUNCTIONS: G17, G18 E G19

Explanation: Selects work plan

The functions **"G17"**, **"G18"** and **"G19"** selects the work plan that the interpolation will be make.



The select work plan function is modal.

Explantion:

G17 - SELECTS XY WORK PLAN
G18 - SELECTS XZ WORK PLAN
G19 - SELECTS YZ WORK PLAN

NOTE: *The G17 plan is the most used to generates profiles and, because of this, it will be used in this Manual.*

14.2 - FUNCTIONS: G500, G53 AND SUPA

Application: Modal and non modal work coordinate system cancellation.

The function **"G500"** has the purpose to cancel the work piece coordinate (G54 to G57 and G505 to G599 function), leaving as reference to work the machine zero coordinate. This function is modal.

The **"G53"**and **"SUPA"** functions aren't modals, therefore has the purpose to cancel the work piece coordinate only at the block where these functions are programmed. Being like this, the next blocks wiil have as reference the last work piece coordinate system activated.

Example:

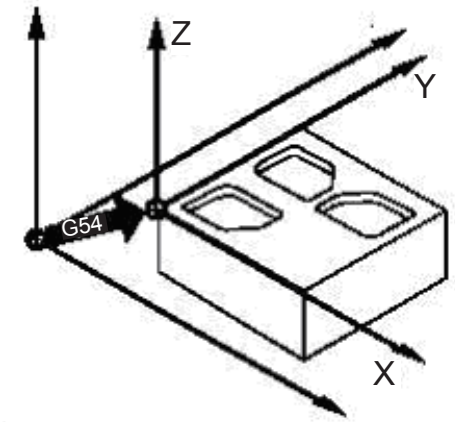
```
G53 G0 Z0 D0;  MOVE THE Z AXIS IN THE Z0 MACHINE COORDINATE.
               (NEAR FROM TOOL CHANGE POINT)
```

14.3 - FUNCTIONS: G54 A G59 E G507 A G599

Explanation: Work coordiante system (WCS)

The work coordinate system defines as zero a given point referenced in the part.

This system can be established by one of the fifty-four functions between **"G54"** to **"G59"** and **"G507"** to **"G599"**.



The referencing values must be inserted in "WORK PIECE REFERENCE" page and represent the distance to each axis from zero machine to zero-part.

Explanation:

G54 - ACTIVATES WORK COORDINATE 1
G55 - ACTIVATES WORK COORDINATE 2
:
G59 - ACTIVATES WORK COORDINATE 6
G507 - ACTIVATES WORK COORDINATE 7
:
G599 - ACTIVATES WORK COORDINATE 99

14.4 - FUNCTIONS: G09 AND G60

Application: Exact position.

G09 and G60 functions are used to execute moves with exacts end points coordinates, as, for example, live corners. With this, at each executed move, the control generates a small axes stop for all axes involved in this move.

The **G09** function isn't modal, therefore it's valid only in blocks where it was programmed. Being like this, next blocks will be executed with the last function of trajectory control activated (G60, G64, G641 or G642)

The **G60** function is default, modal and cancel the G64, G641 and G642 functions.

14.5 - FUNCTIONS: G64, G641, G642

Application: Continuous trajectory control

The **"G64"**, **"G641"** and **"G642"** functions are used to the control some blocks ahead ("Look ahead") and generates small chanfers or roundings in the transition corners created between moves.

These codes do the machine work with a constant trajectory and keep the axes move in a continuous way, without stop between blocks executions.

Codes explanation:

G64 - CHAMFER IN CORNERS.
G641 - ROUNDING IN CORNERS.
G642 - CORNERS IN SPLINE FORM.

These functions are modals and cancel the G60 function.

14.6 - FUNCTION: G70

Application: Inches programming.

A G70 block in the program beginning instructs the control to use inches values to axes moves, feedrates and corrections.

The G70 function is modal.

14.7 - FUNÇÃO: G71

Application: Milimiters Programming

A G71 block in the program beginning instructs the control to use milimiters values to axes moves, feedrates and corrections.

The G71 function is modal.

14.8 - FUNÇÃO: G90

Explanation: Absolute coordinates

In the absolute coordinate system, the axes' positions are measured from the predefined zero-part, and we must always inform the position to where the tool must be moved to program in this system.
This function is modal.

Syntax:

G90 - MODAL
X=AC(...) Y=AC(...) Z=AC(...) - NOT MODAL

14.9 - FUNCTION: G91

Explanation: Incremental coordinates

In the incremental coordinate system, axes' positions are measured from the position previously established, and we must inform the distance to be traveled by the tool as from the current position.

This function modal.

Syntax:

G91 - MODAL
X=IC(...) Y=IC(...) Z=IC(...) - NOT MODAL

14.10 - FUNÇÃO: G94

Explanation: Feed in milimeters/inches per minute

The feedrate it's declared with the **"F"** function.

The G94 function is modal, and is activated automatically when the machine is turned on.

14.11 - FUNÇÃO: G95

Explanation: Feed in mm/revolution ou inches/revolution

The feedrate it's declared with the **"F"** function, this function is, usually, used in lathes.

The G95 function is modal.

15 - REPETITIONS AND JUMPS

15.1 - FUNCTION: REPEAT

Application: To repeat a block or a program section.

The REPEAT function is used to repeat a specific block or a program part. In the second case, can be a sub routine.

"LABEL" is the word address to mark the beginning and the end of the deviation, or the block to be repeated. The word that represents the "LABEL" should come followed of two points (**:**).

REPEATB is the repetition sentence used to execute again only the program block where is the desired "LABEL". The **"P"** function determines the repetition numbers (n).

REPEAT is the repetition sentence used to execute again a program part. In the same block is programmed the "LABEL" where the repetition will start and the "LABEL" where the repetition will finish. The **"P"** function determines the repetition numbers (n).

Example 1: REPEATB

```
  :
N100 G0 X10 Y20
N110 Z2
N120 HELIX: G2 X=IC(0) Y=IC(0) Z-20 I-10 J0 TURN=11 F600
N130 G1 X0 F600
N140 G0 Z2
N150 G0 X80 Y45
N160 REPEATB HELIX
N170 G1 X70
N180 G0 Z2
  :
```

In the above example, the "HELIX" word is the "LABEL". After executes the block N150, the machine will execute again the block where is the HELIX word (N120) and, after execute this block, the executon will continue starting from the block N170.

Example 2:

```
  :
N10 G0 X-10 Y-10
N20 DEEPEN: G1 Z=IC(-2)F100;
N30 G1 X0 Y0
N40 X100
N50 Y100
N60 X0
N70 Y0
N80 X-10 Y-10
N90 REPEAT DEEPEN P4
 :
```

In the example before, the word “DEEPEN” is the “LABEL” . The machine will execute again since the block where is the word “DEEPEN”(N20) until the block before the REPEAT function (N80) 4 times.

Example 3:

```
  :
N230 T07; DRILL D10 MM
N240 M6
N250 G54 D1 S1650 M3 G64 CFIN
N260 G0 X35 Y60
N270 Z10
N280 F350
N290 MCALL CYCLE82(5,0,2,-20,,0.2)
N300 START: X35 Y60
N310 Y115
N320 X145
N330 END: Y60
N340 MCALL
N350 G53 G0 Z-110 D0
N360 T12; D20 MM
N370 M6
N380 G54 D1 S2800 M3 G64 CFIN
N390 G0 X35 Y60
N400 Z10
N410 F600
N420 MCALL CYCLE82(5,0,-6,0,0.5)
N430 REPEAT START END
N440 MCALL
  :
```

In the example above, the word “START” is the initial “LABEL” and the word “END” is the final “LABEL”. The machine will execute again the instructions from the “START” block (N300) to the “END” block(N330), the which, in this case, indicate the positions where the pockets will be done.

15.2 - FUNCTION: GOTO

Application: Program deviation.

When it’s necessary to program an execution desviation (a jump), to a specific part of the same program, it’s used the GOTO function with the desired “LABEL” (address).

“LABEL” is the word address to mark the desviation point. The word that represents the “LABEL” should come followed of two points (**:**).

GOTOB function makes the desviation to a block before the current block.

GOTOF function makes the desviation to a block after the current block.

Example 1:

```
    N240 G53 G0 Z-110 D0
┌── N250 GOTOF AHEAD
│   N260 T03; REAMING
│     :
│     :
└─► N350 AHEAD:
    N360 T04; MILLING
       :
```

In the example above, the machine will jump from the block N250 (function “GOTOF”) to the block N350 (AHEAD LABEL).

Example 2:

```
┌─► N10 RESTART:
│   N20 G500 G0 X0 Y0
│   N30 X-560
│   N40 X0 Y-406
│   N50 X-560
└── N60 GOTOB RESTART
```

In the example above, the machine will execute from the block N10 (LABEL “RESTART”) until the block N60 (function GOTOB) and, after that, it will enter in “LOOPING”, it means, will return to the block N10 and will execute until the block N60 again, and satys doing this until the execution being stopped by the operator.

16 - SUBPROGRAM CALL

The subprogram call feature can be used when the machining of a sequence of operations must be repeated several times.

The subprogram is called by the name. Ex: “SUBPRO1.SPF”

The subprogram names should follow the rules below:

– All characterses should be letters, or should start for the letter **“L”** and to finish with numbers, or then, the first two characterses should be letters and the others can be numbers.

– Use the maximun of 31 characterses

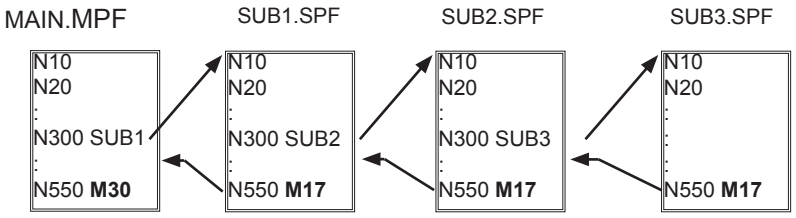
– Don’t use separation characterses (, - / “ ”)

– Extension “**.SPF”**
EX: L120.SPF

NOTES: *The main programs should be follow the same rules above, but should have the* ***“.MPF”*** *extension.*

The subprogram call is done informing the subprogram name in the main program. The subprograms can be called by the main program and, also, by other subprogram, with this, from the main program you can call until 11 subprogram in sequence.

Example 1:



NOTES: *In the end of subprogram should be programmed the* ***“M17”*** *function. In the end of main program should be programmed the* ***“M30”*** *function.*

The “P” function determines how many times the subprogram will be executed. This function should be programmed together with the subprogram call.

Example 2:

MAIN.MPF

G0 X10 Y10 Z0
L120 P3
G90 G0 Z100
M30

L120.SPF

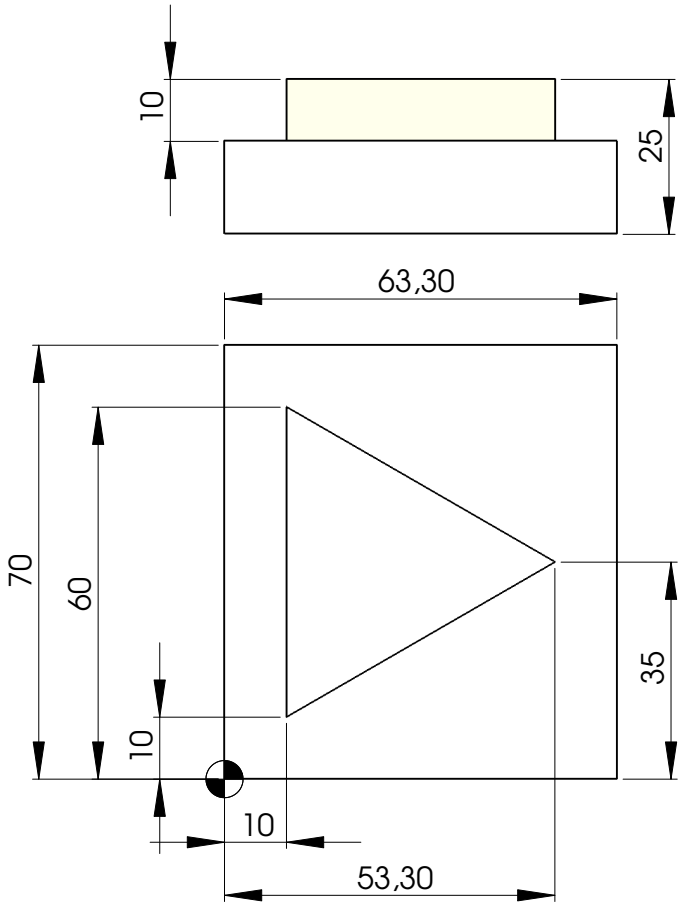
G91 G1 X50 Y50 F50
X50 Y-50
M17

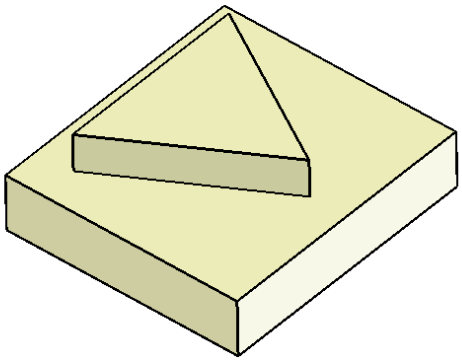
Call the path **L120.SPF**, 3 times

Example 3:

Main program: **PROFILE.MPF**
G17 G71 G90 G94
G53 G0 Z-110 D0
T01
M6
G54 D01 G64 CFIN
S2000 M3
G0 X0 Y0 Z10
G1 Z0 F300
TRIANGLE P4
G0 Z10
G53 G0 Z-110 D0 M5
M30

Subprogram **TRIANGLE.SPF**
G1 Z=IC(-2.5) F100
G41 X10 Y10 F200
Y60
X53.3 Y35
X10 Y10
G40 X0 Y0
M17





In the example the main program PROFILE.MPF called and executed the subprogram TRIANGLE.SPF 4 times, so that, the tool will arrive to the depth of 10 mm.

17 - MCALL - SUBPROGRAM AND CYCLE MODAL CALL.

This function is used to become modals the cycles and subprograms that are programmed together this function, this way, the cycle or subprogram will be executed to each new informed coordinate below. The execution will continue until the MCALL function to be programmed alone in a block.

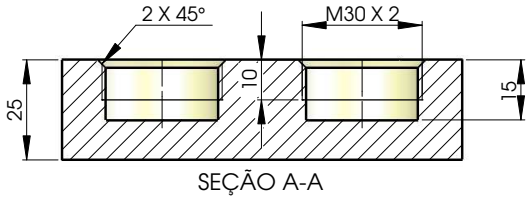
Syntax:

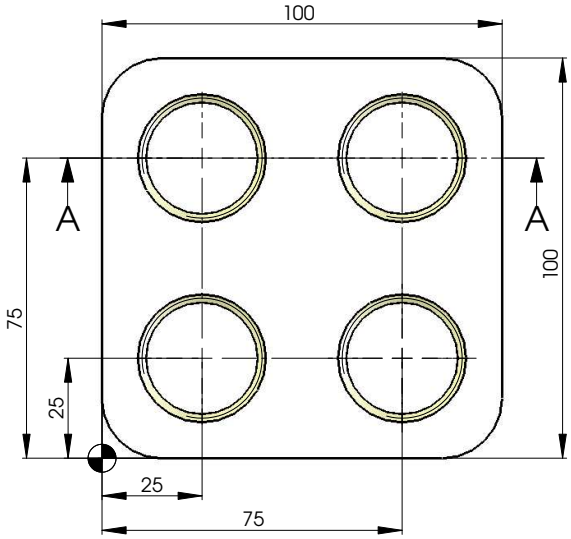
MCALL CYCLE83 (5,0,2,-100, ,-20, ,5,1,2,1,0) - BECOME MODAL THE CYCLE83.
X__ Y__ - APPROACH IN THIS COORDINATE AND EXECUTE THE CYCLE83 (FIRST HOLE)
X__ Y__ - APPROACH IN THIS COORDINATE AND EXECUTE THE CYCLE83 (SECOND HOLE)
:
MCALL - DESACTIVES THE CYCLE83

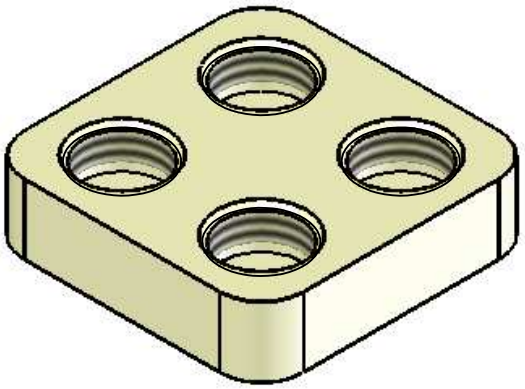
OR

MCALL PERFIL_EXT - BECOME MODAL THE SUBPROGRAM “PERFIL_EXT”
X__ Y__ - APPROACH IN THIS COORDINATE AND EXECUTE THE SUBPROGRAM “PERFIL_EXT”
X__ Y__ -APPROACH IN THIS COORDINATE AND EXECUTE THE SUBPROGRAM “PERFIL_EXT”
:
MCALL - DESACTIVES THE SUBPROGRAM “PERFIL_EXT”

Example:







PROGRAM “EXE_MCALL.MPF”

```
G17 G71 G90 G94
G53 G0 Z-110 D0
T01; DRILL D28 MM
M6
G54 D01 S1500 M3 G64 CFIN
G0 X25 Y25
Z10
F300
MCALL CYCLE82(5,0,2,-15)
X25 Y25
X75
Y75
X25
MCALL
G53 G0 Z-110 D0 M5
T02; THREAD M30X2 - THREAD MILL D16
M6
G54 D01 S2800 M3 G64 CFIN
G0 X25 Y25
Z10
MCALL THREAD_M30X2
X25 Y25
X75
Y75
X25
MCALL
G53 G0 Z-110 D0 M5
M30
```

SUBPROGRAM “THREAD_M30X2.SPF”

```
G1 Z-10 F1000
G91 X3.5 Y-3.5
G3 X3.5 Y3.5 Z0.25 CR=3.5
G3 X0 Y0 I-7 J0 Z2
G3 X-3.5 Y3.5 Z0.25 CR=3.5
G1 X-3.5 Y-3.5
G90 G0 Z10
M17
```

18 - OFFN FUNCTION.

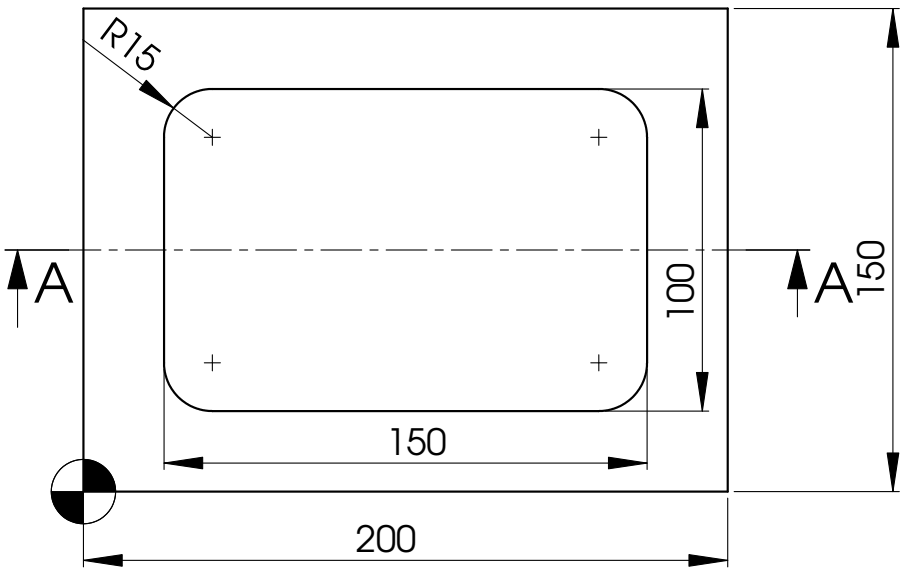
The OFFN function is used in contour milling operation when it's necessary perform moves with a offset value from the original profile. This function can be used only in programs where the tool radius compensation is active.

Syntax:

OFFN=___ - DETERMINES THE OFFSET VALUE TO BE APLICATED ON THE ORIGINAL PROFILE.
OFFN=0 - DESACTIVES THE OFFN FUNCTION.

Exemplo:

```
N10 G17 G71 G90 G94
N20 G53 G0 Z0 D0
N30 T01; MILL D16
N40 M6
N50 G54 D01 S3200 M3 G64 CFIN
N60 G0 X-13 Y-13
N70 Z5
N80 G1 Z-5 F1000
N90 OFFN=12.5
N100 START: G41 G1 X25 Y25
N110 G1 X25 Y125 RND=15
N120 G1 X175 Y125 RND=15
N130 G1 X175 Y25 RND=15
N140 G1 X25 Y25 RND=15
N150 G1 X25 Y45
N160 END: G40 G1 X-13 Y-13
N170 OFFN=0.2
N180 REPEAT START END
N190 OFFN=0
N200 REPEAT START END
N210 G0 Z10
N220 G53 G0 Z0 D0
N230 M30
```



19 - CYCLES.

The cycles are macros developed for the CNC builder with the purpouse of become easier the programming of communs operations as: drilling, rigid tapping, boring, facing, thread milling, etc.

Syntax:

CYCLEnn(aa , aa , ... , aa)
HOLESnn(aa , aa , ... , aa)
LONGHOLE(aa , aa , ... , aa)
POCKETnn(aa , aa , ... , aa)
SLOTnn(aa , aa , ... , aa)

where:
CYCLE..., HOLES..., LONGHOLE, POCKET..., SLOT... - cycles name.
nn - number of cycle identification. Examples: CYCLE82, CYCLE90, HOLES2, etc
aa - cycle arguments

Example: CYCLE82(5,0,2,0,0.5)

In the Siemens 828D control the cycles can be programed two ways: Using the interactive screen or programming manually the complete cycle in the edition screen.

In the interactive screen, some data cycles stay hidden because they are equivalent other visible data cycles.

Independently there are been occult data or not, when the cycle be inserted in the program, the syntax it will be complete.

In the next chapters to proceed will be shown like acess the support screens of each one of the cycles.

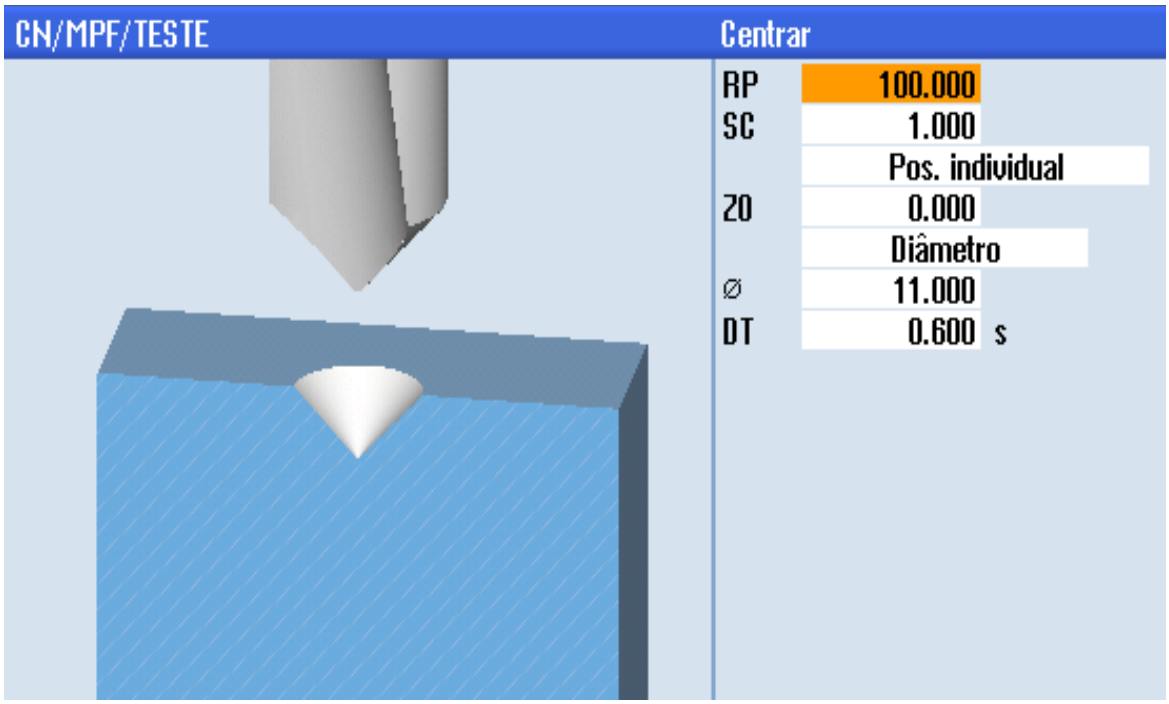
19.1 - CYCLE81

Function: Drilling, centering

This cycle is used to make simple holes, like center holes, small holes, etc...

To acess the page of programming of the CYCLE81 it's necessary to follow the following steps:

- Press the **[Drilling]** softkey
- Press the **[Centering]** softkey
- Insert the data in the blanks
- Press the **[Accept]** softkey



The data to be inputed are the following ones:

RP	Coordinate Z of return of the tool after the end of the cycle (absolute)
SC	Safe distance (distance to approach - without sign)
3nd Blank	Through the **[SELECT]** key choose the follow options: **- Single Position** (just one hole) - **Position Pattern (MCALL)** (it accomplishes several holes with the same depth)
Z0	Coordinate "Z" of the start point
5th Blank	Through the [SELECT] key choose the follow options: **Diameter** (determinate the end diameter of drilling) **Tip** (determinate Final drilling depth)
Ø	End diameter of drilling.
Z1	Final drilling depth. Choose the option Absolute or Incremental Through the **[SELECT]** key
DT	Dwell time at final drilling depth. Choose th option Time or RPM Through the **[SELECT]** key

After press the **[Accept]** Softkey, the cycle is inserted in the program according to follow example:

CYCLE81 (_____)

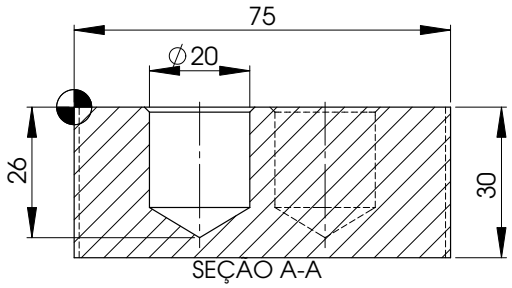
NOTE:

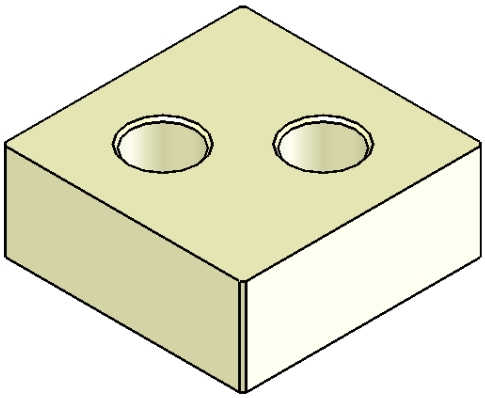
- Feed and RPM must be programmed in a single block, BEFORE the cycle 81.

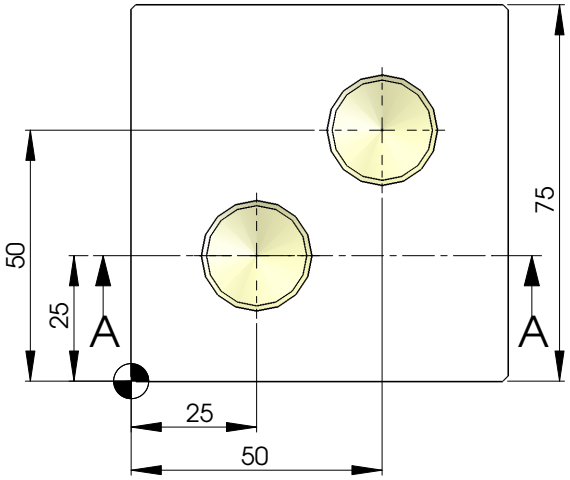
- The data roughing (feed and rpm) must be programmed before, in a single block. It's necessary to program just a value for the end of the drill, that is to say, programming the "Z1" or "E".

Some data can be hidden in the block. This data receive the 0 value.

```
WORKPIECE(,,"","BOX",112,0,-30,-80,0,0,75,75)
G17 G71 G90 G94
G53 G0 Z-110 D0
T01; DRILL D20 MM
M6
G54 D01 S1800 M3
G0 X25 Y25
Z10
F100
MCALL CYCLE81(5,0,2,-26,,0,0,0,22)
X25 Y25
X50 Y50
MCALL
G53 G0 Z-110 D0 M5
M30
```







19.2 - CYCLE82

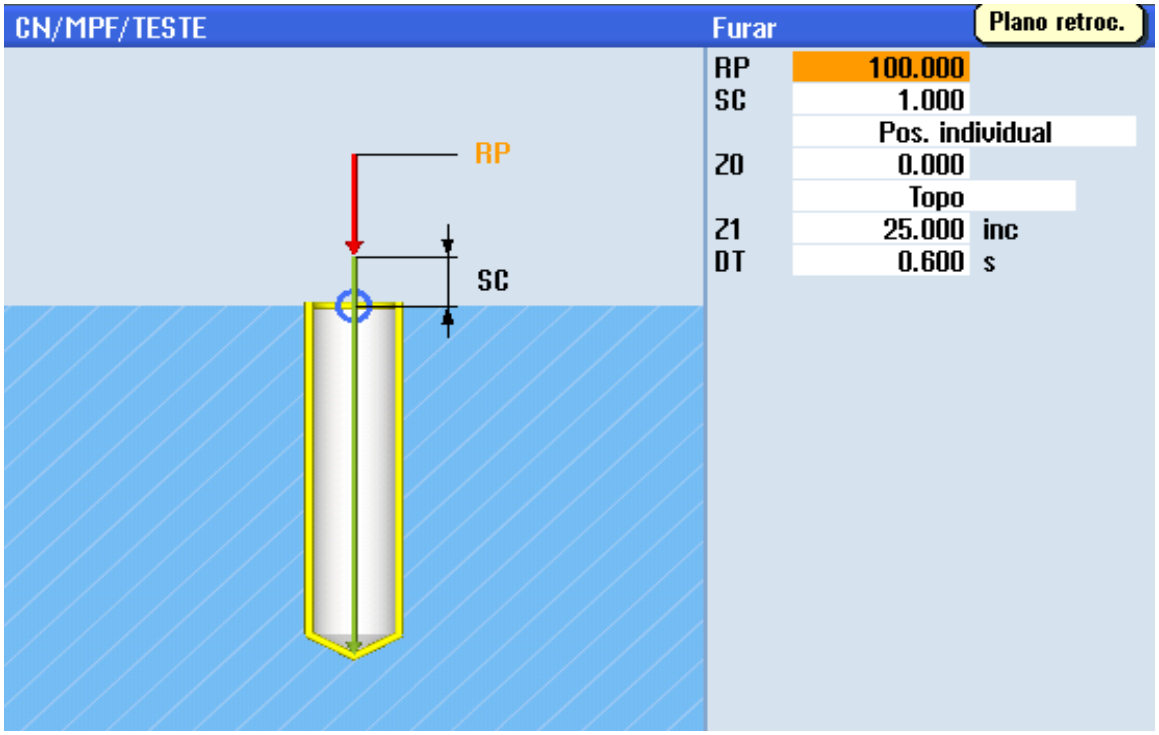
Function: Drilling, counterboring

This cycle is used to make simple holes, like counterboring.

The tool drills with the spindle speed and feedrate programmed at the final drilling depth. A dwell time can be programed to elapse when the final drilling depth has been reached.

To acess the page of programming CYCLE82 it's necessary to follow these steps:

- Press the **[Drilling]** softkey
- Press the **[Drilling Reaming]** softkey
- Insert the data in the blanks
- Press the **[Accept]** softkey



The data to be inputed are the following ones:

RP	Coordinate Z of return of the tool after the end of the cycle (absolute)
SC	Safe distance (distance to approach - without sign)
3rd Blank	Through the **[SELECT]** key choose the follow options: **- Single Position** (just one hole) - **Position Pattern (MCALL)** (it accomplishes several holes with the same depth)
Z0	Coordinate "Z" of the start point
5th Blank	Through the [SELECT] key choose the follow options: **Diameter** (determinate the end diameter of drilling) **Tip** (determinate Final drilling depth)

Z1	Final drilling depth. Choose the option Absolute or Incremental Through the **[SELECT]** key
DT	Dwell time at final drilling depth. Choose th option Time or RPM Through the **[SELECT]** key

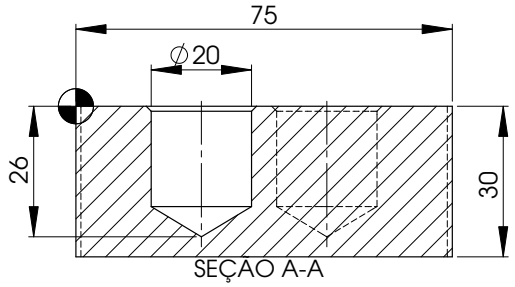
After pess the **[Accept]** Softkey, the cycle is inserted in the program according to follow example:

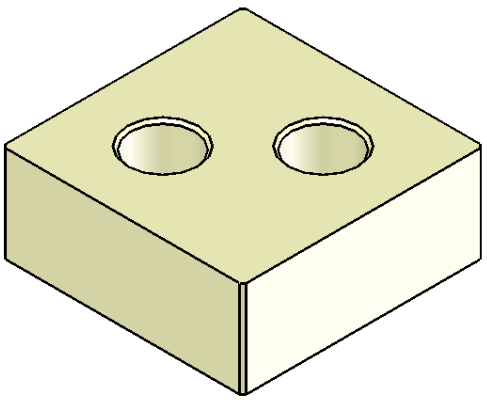
CYCLE82 (_____)

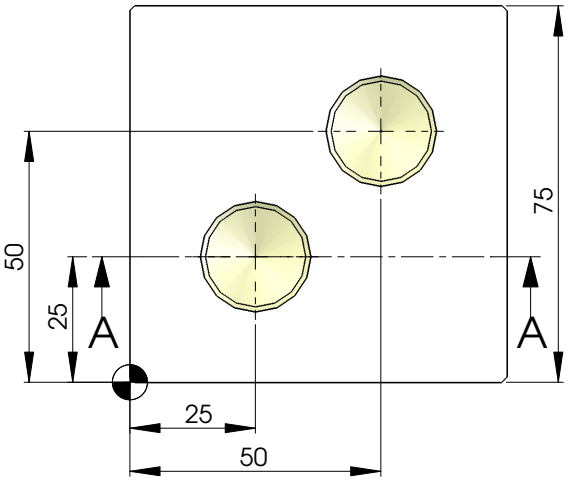
NOTE:
- Feed and RPM must be programmed in a single block, BEFORE the cycle 81.
- It's necessary to program just a value for the end of the drill, that is to say, programming the "Z1" or "E".
Some data can be hidden in the block. This data receive the 0 value.

Example:

```
WORKPIECE(,,"","BOX",112,0,-30,-80,0,0,75,75)
G17 G71 G90 G94
G53 G0 Z-110 D0
T01; DRILL D20 MM
M6
G54 D01 S1800 M3
G0 X25 Y25
Z10
F100
N100 MCALL CYCLE82(5,0,2,-26,,0.5,10,1,12)
X25 Y25
X50 Y50
MCALL
G53 G0 Z-110 D0 M5
M30
```







19.3 - CYCLE83

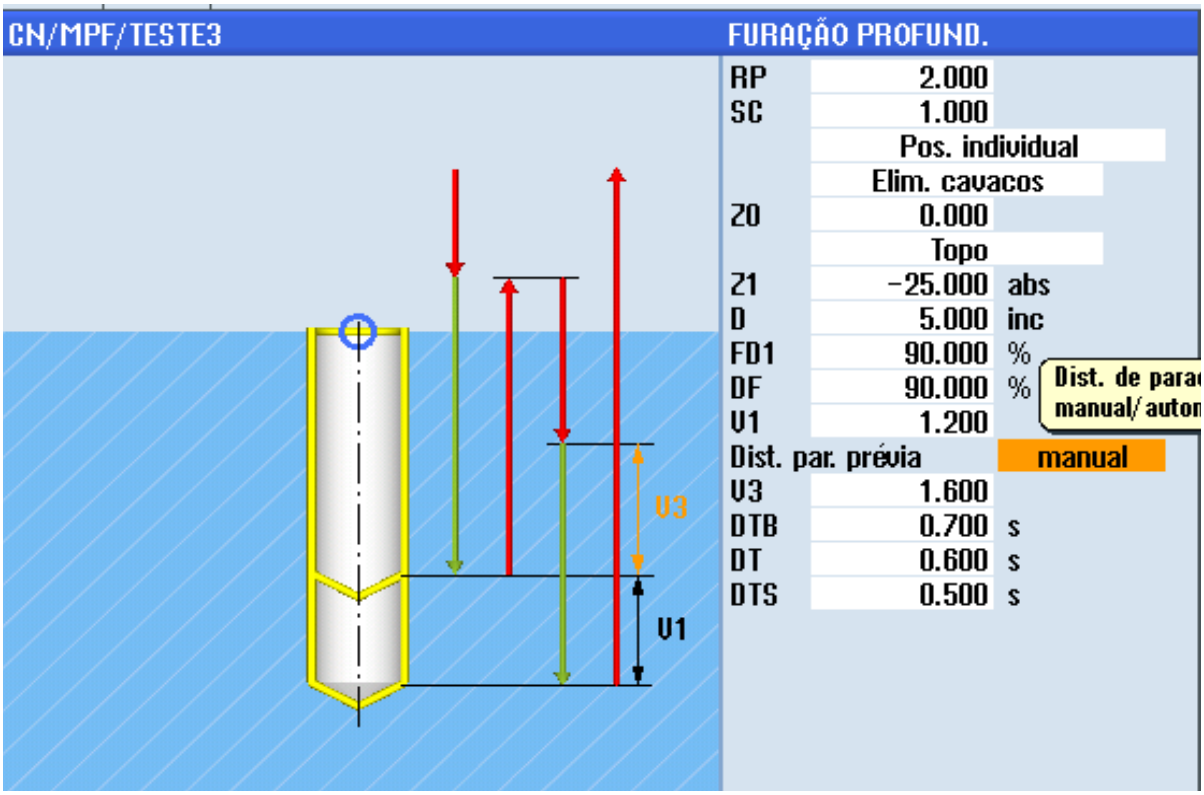
Function: Deep hole

Deep hole drilling is performed with a depth infeed of a maximum definable depth executed several times, increasing gradually until the final drilling depth is reached.

The drill can either be retracted to the reference plan + safety clearance after every infeed depth for swarf removal or retracted a bit for chip breaking.

To acess the page of programming CYCLE83 it's necessary to follow these steps:

- Press the **[Drilling]** softkey
- Press the **[Deep Hole Drilling]** softkey
- Insert the data in the blanks
- Press the **[Accept]** softkey



The data to be inputed are the following ones:

RP	Coordinate Z of return of the tool after the end of the cycle (absolute)
SC	Safe distance (distance to approach - without sign)
3rd Blank	Through the **[SELECT]** key choose the follow options: **- Single Position** (just one hole) - **Position Pattern (MCALL)** (it accomplishes several holes with the same depth)

4th Blank	Through the **[SELECT]** key choose the follow options: - **Chip breaking** - **Chip removal**
Z0	Coordinate “Z” of the start point
6th Blank	Through the **[SELECT]** key choose the follow options: - **Top** - **Tip**
Ø	End Diameter.
Z1	Final drilling depth. Choose the option Absolute or Incremental Through the **[SELECT]** key.
D	First drilling depth. Choose the option Absolute or Incremental Through the **[SELECT]** key.
FD1	Feedrate factor for first drilling depth
DF	Degression. Choose the option degression or mm Through the **[SELECT]** key.
V1	Minimum drilling depth.
V2	Retraction.
Dist. par. prévia	Programmable limit distance on re-insertion in hole. Through the **[SELECT]** key choose the follow options: - **Manual** (programmable value applies) - **Automatic** (automatic calculation)
V3	Programmable value applied on re-insertion in hole
DTB	Dwell time at drilling depth (chip breaking). Choose the option second or RPM Through the **[SELECT]** key.
DT	Dwell time at final drilling depth. Choose the option second or RPM Through the **[SELECT]** key.
DTS	Dwell time at starting point and for swarf removal. Choose the option second or RPM Through the **[SELECT]** key.

After press the **[Accept]** Softkey, the cycle is inserted in the program according to follow example:

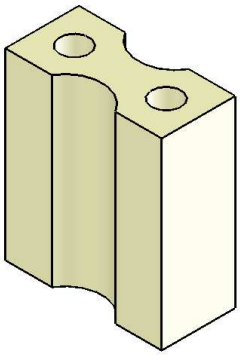
CYCLE83 (_____)

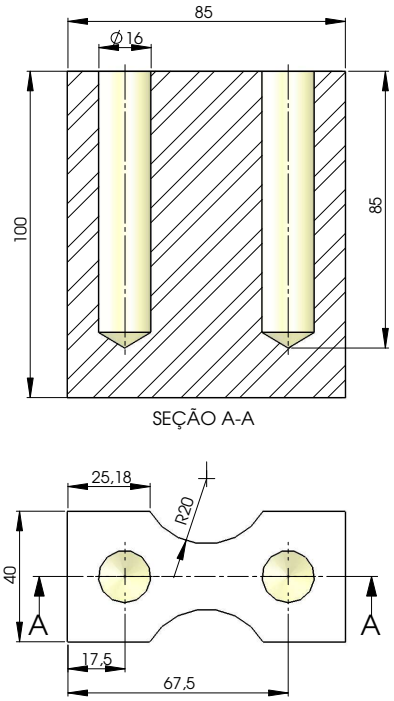
NOTE:

- *Feed and RPM must be programmed in a single block, BEFORE the cycle 83.*
- *It’s necessary to program just a value for the end of the drill, that is to say, programming the DP or DPR.*
- *It’s necessary to program just one value for the first depth (Absolute or Incremental)*
- *Some data can be hidden in the block. This data receive the 0 value.*

Example:

```
WORKPIECE(,,"","BOX",112,0,-30,-80,0,0,75,75)
G17 G71 G90 G94
G53 G0 Z-110 D0
T15; DRILL D16 MM
M6
G54 D01 S2000 M3
G0 X17.5 Y20
Z7
F200
MCALL CYCLE83(5,0,2,-85,,-20,,90,1,0,1.2,1.4,0.6,1.6,10,1,11221112)
X17.5 Y20
X67.5
MCALL
G53 G0 Z-110 D0 M5
M30
```





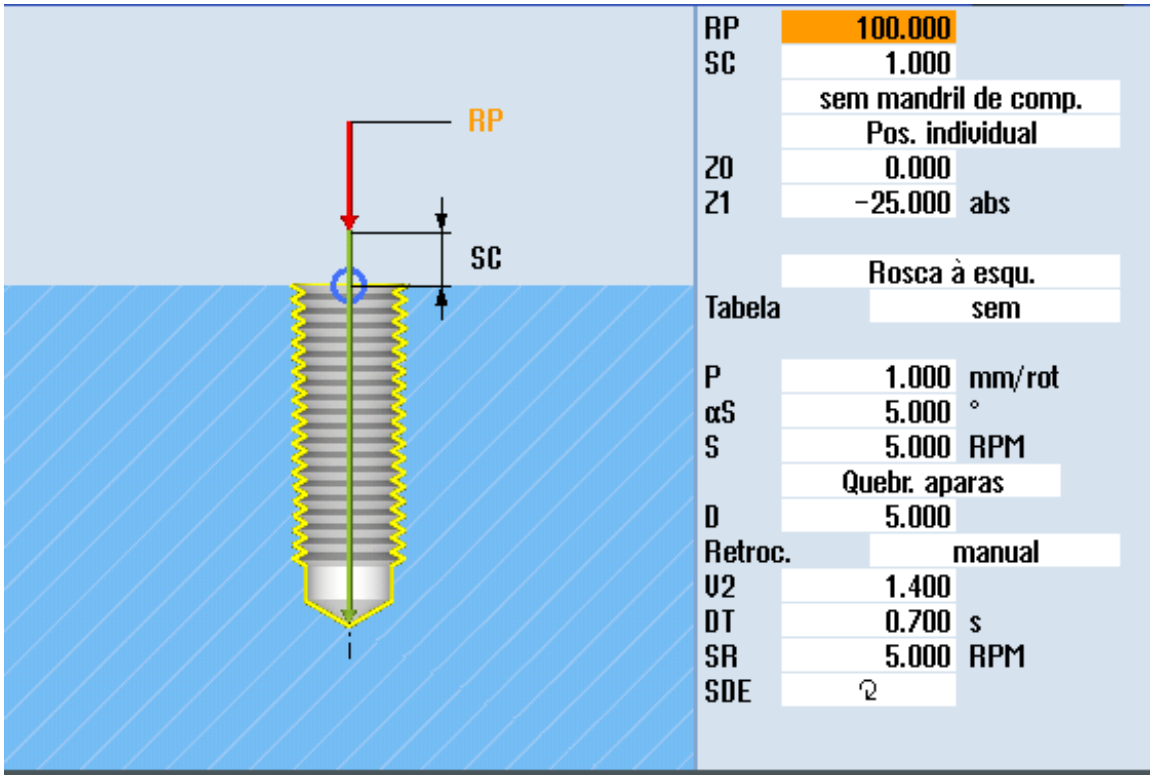
19.4 - CYCLE84

Function: Rigid tapping

The tool drills with the spindle speed and feedrate programmed at the thread depth. With cycle CYCLE84 you can perform rigid tapping operations..

To acess the page of programming CYCLE84 it's necessary to follow these steps:

- Press the **[Drilling]** softkey
- Press the **[Thread]** softkey
- Insert the data in the blanks
- Press the **[Accept]** softkey



The data to be inputed are the following ones:

RP	Coordinate Z of return of the tool after the end of the cycle (absolute)
SC	Safe distance (distance to approach - without sign)
3rd Blank	Through the **[SELECT]** key choose the follow options: - **With compensating chuck** **- Without compensating chuck**
4th Blank	Through the **[SELECT]** key choose the follow options: **- Single Positions** (just one hole) - **Position Pattern (MCALL)** (it accomplishes several holes with the same depth)
Z0	Coordinate "Z" of the start point

Z1	Final drilling depth. Choose the option Absolute or Incremental Through the **[SELECT]** key.
7th Blank	Through the **[SELECT]** key choose the direction of rotation in the thread: **- Right** **- Left**
Table	Through the **[SELECT]** key choose the follow options: - **ISO métric** **- Whitworth BSW** **- Whitworth BSP** **- UNC**
Select	Through the **[SELECT]** key choose the Pitch as thread size.
P	Pitch. Through the **[SELECT]** key choose mm or withworth.
αS	Spindle position for oriented spindle stop in the cycle (in degrees)
S	Speed for tapping
13th Blank	Through the **[SELECT]** key choose the follow options: - **1 depth** - **Chip breaking**
DT	Dwell time at thread depth
SR	Speed for retraction.
SDE	Through the **[SELECT]** key choose the direction of rotation after the end of cycle.

After press the **[Accept]** Softkey, the cycle is inserted in the program according to follow example:

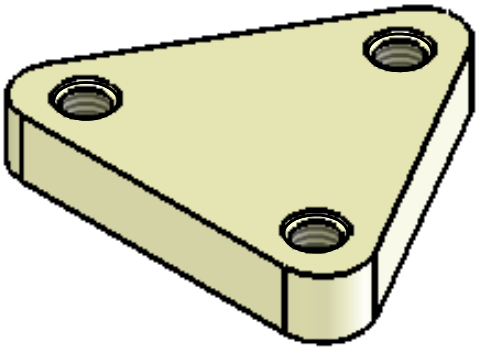
CYCLE84 (_____)

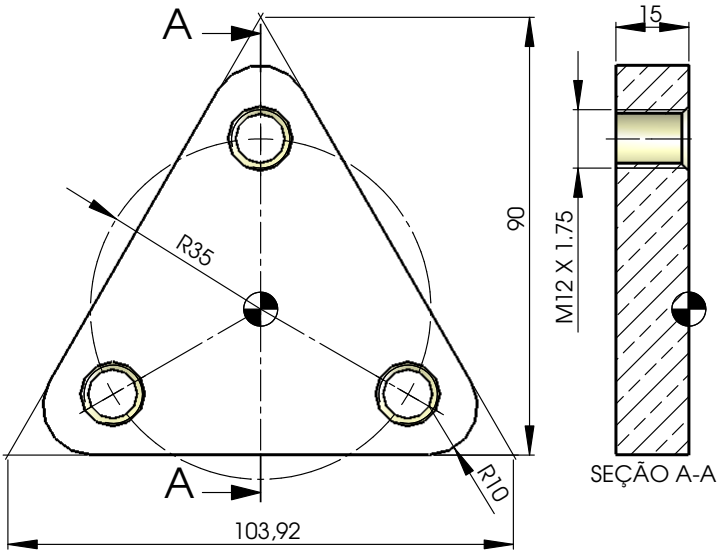
NOTE:

- *RPM must be programmed in a single block, BEFORE the cycle 84.*
- *It's necessary to program just a value for the end of the drill, that is to say, programming the "Z1" or "E".*
- *Some data can be hidden in the block. This data receive the 0 value.*

Example:

```
WORKPIECE(,,"","BOX",112,0,-15,-80,-65,-65,65,65)
G17 G71 G90 G94
G53 G0 Z-110 D0
T20; TAP M12X1.75
M6
G54 D01 S500 M3
G0 X0 Y35
Z5
MCALL CYCLE84(5,0,2,-18,,0.7,3,,1.75,5,500,5,0,1,0,1,5,1,,,,,1001,2001002)
RP=35 AP=90
AP=210
AP=330
MCALL
G53 G0 Z-110 D0 M5
M30
```





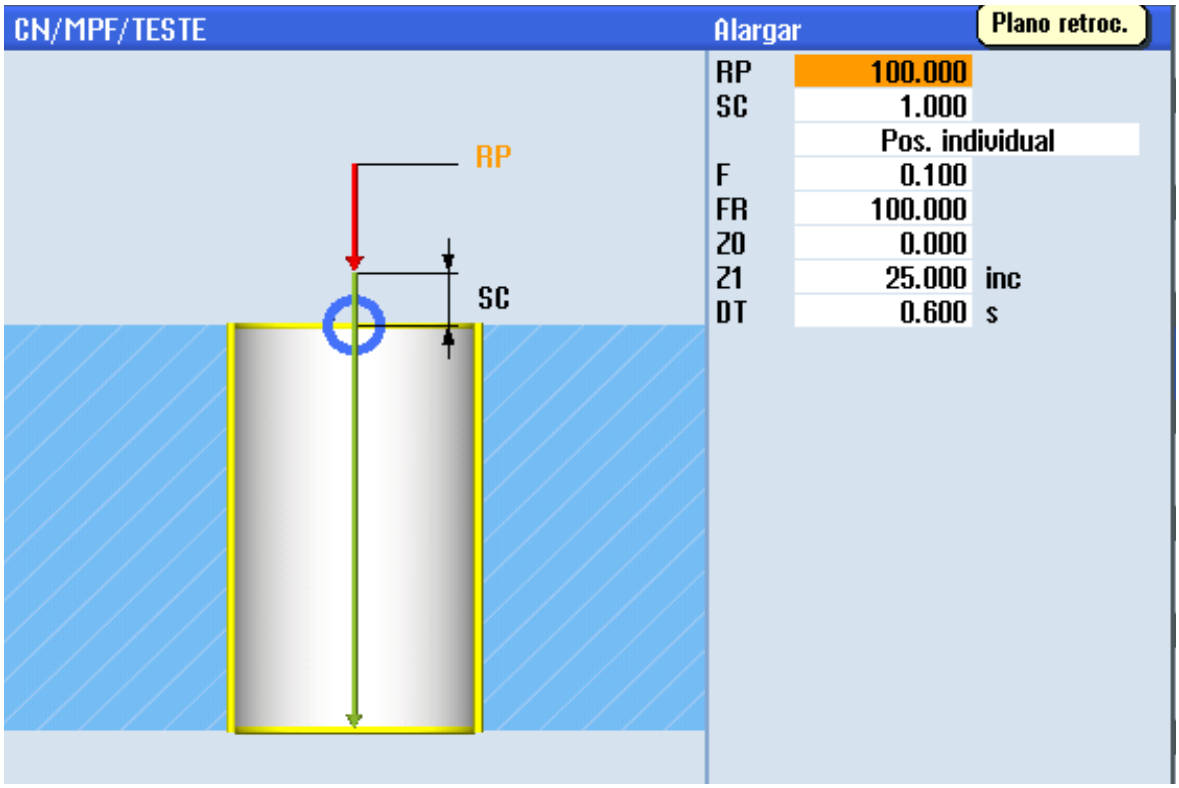
19.5 - CYCLE85

Function: Boring.

The tool drills at the programmed spindle speed and feedrate to the programmed final drilling depth.

To acess the page of programming of the CYCLE85 it's necessary to follow the following steps:

- Press the **[Drilling]** softkey
- Press the **[Drilling Boring]** softkey
- Insert the data in the blanks
- Press the **[Accept]** softkey



The data to be inputed are the following ones:

RP	Coordinate Z of return of the tool after the end of the cycle (absolute)
SC	Safe distance (distance to approach - without sign)
13th Blank	Through the **[SELECT]** key choose the follow options: **- Single Position** (just one hole) - **Position Pattern (MCALL)** (it accomplishes several holes with the same depth)
F	Feedrate
FR	Retraction feedrate
Z0	Coordinate "Z" of the start point

Z1	Final drilling depth. Choose the option Absolute or Incremental Through the **[SELECT]** key.
DT	Dwell time at boring depth.

After pess the **[Accept]** Softkey, the cycle is inserted in the program according to follow example:

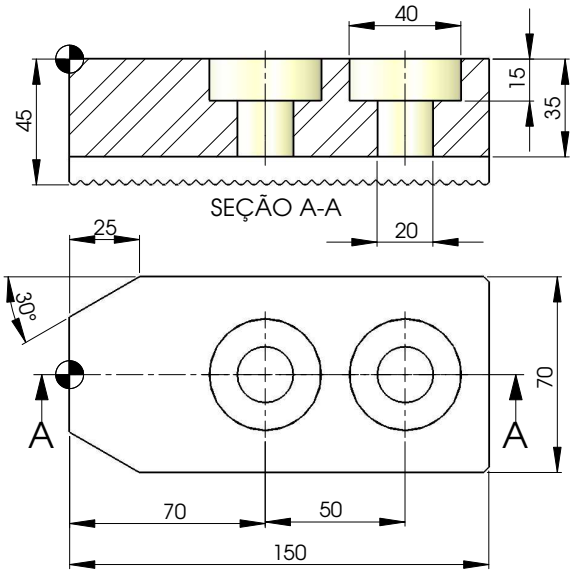
CYCLE85 (_____)

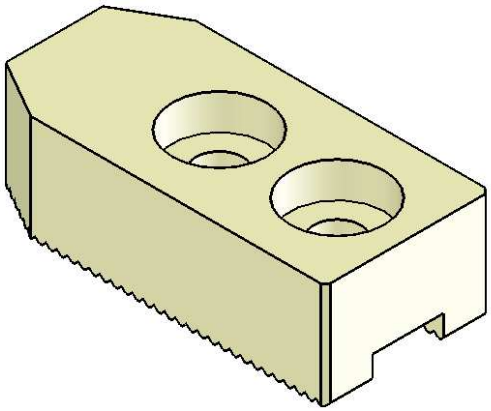
NOTE:

- *RPM must be programmed in a single block, BEFORE the cycle 85.*
- *It's necessary to program just a value for the end of the drill, that is to say, programming the "Z1" or "E".*
- *Some data can be hidden in the block. This data receive the 0 value.*

Example:

```
  :
G53 G0 Z-110 D0
T04
M6
G54 D1 S920 M3
G0 X70 Y0
Z15
MCALL CYCLE85 (5,0,2,-15, ,1,100,500)
X70 Y0
X120
MCALL
G53 G0 Z-110 D0 M5
  :
```





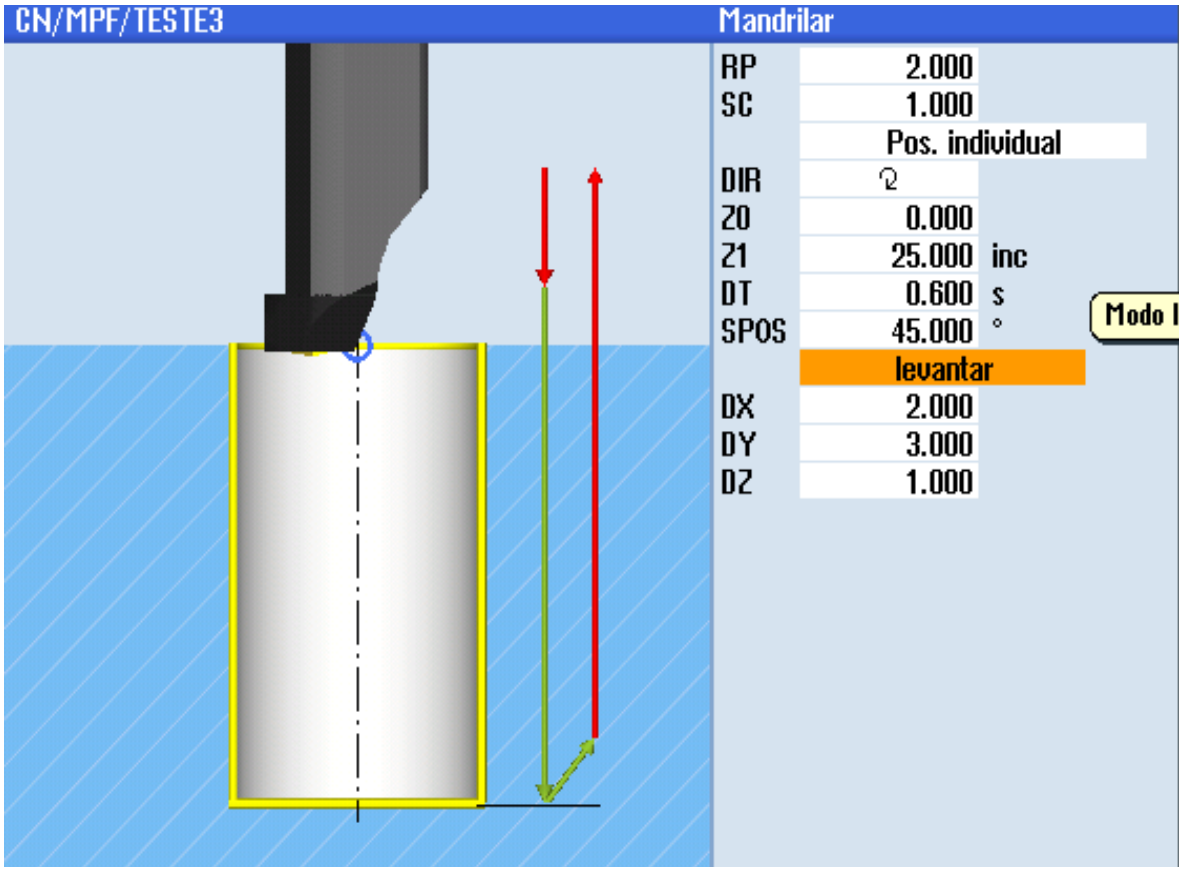
19.6 - CYCLE86

Function: Boring with oriented spindle stop

The tool drills with the spindle speed and feedrate programmed final drilling depth. With this Boring, oriented spindle stop is activated with once the drilling depth has been reached.

To acess the page of programming CYCLE86 it's necessary to follow these steps:

- Press the **[Drilling]** softkey
- Press the **[Boring]** softkey
- Insert the data in the blanks
- Press the **[Accept]** softkey



The data to be inputed are the following ones:

RP	Coordinate Z of return of the tool after the end of the cycle (absolute)
SC	Safe distance (distance to approach - without sign)
3rd Blank	Through the **[SELECT]** key choose the follow options: **- Singlel Position** (just one hole) - **Position Pattern (MCALL)** (it accomplishes several holes with the same depth)
DIR	Through the **[SELECT]** key choose the follow options: - **Direction of rotation - Clock Wise** - **Direction of rotation - Counter Clock Wise**

Z0	Coordinate “Z” of the start point
Z1	Final drilling depth. Choose the option Absolute or Incremental Through the **[SELECT]** key.
DT	Dwell time at boring depth.
SPOS	Spindle position for oriented spindle stop in the cycle (in degrees)
9th BLANK	Through the **[SELECT]** key choose the follow options: - **Retraction to the reference** (up to three axes with G0) - **Retraction to the reference** (up the “Z” axes with G0)
DX	Retraction path in X plan
DY	Retraction path in Y plan
DZ	Retraction path in Z plan

After press the **[Accept]** Softkey, the cycle is inserted in the program according to follow example:

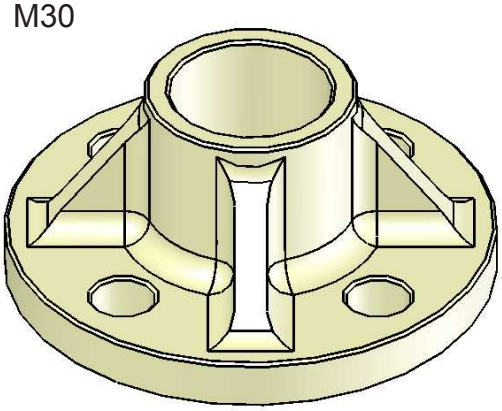
CYCLE86 (_____)

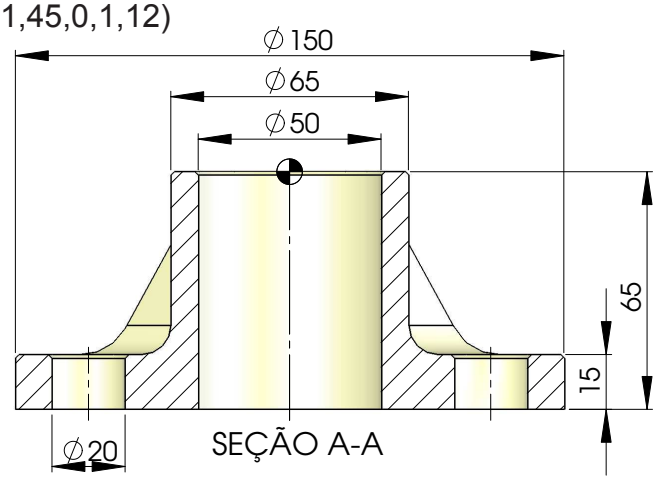
NOTE:

- *Feed and RPM must be programmed in a single block, BEFORE the cycle 86.*
- *It’s necessary to program just a value for the end of the drill, that is to say, programming the “Z1” or “E”.*
- *Some data can be hidden in the block. This data receive the 0 value.*

Example:

```
WORKPIECE(,,"","BOX",112,0,-35,-80,0,-35,150,35)
G17 G71 G90 G94
G53 G0 Z-110 D0
T06
M6
G54 D01 S800 M3
G0 X0 Y0
Z10
F300
MCALL CYCLE86(5,0,2,-15,,0.6,3,2,2,1,45,0,1,12)
G53 G0 Z-110 D0 M5
M30
```





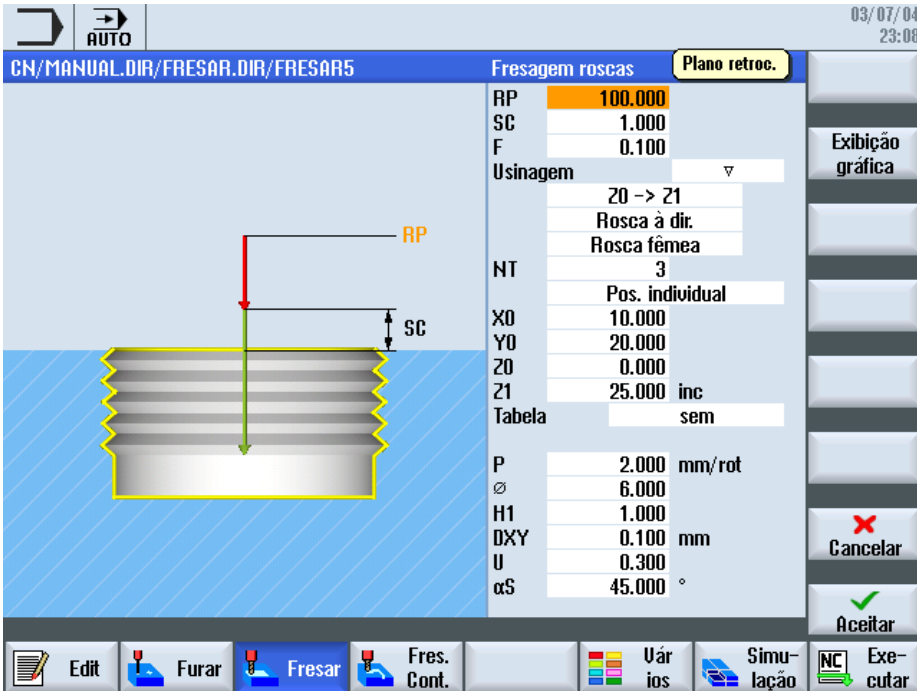
19.7 - CYCLE70

Function: Thread Milling

With this cycle it's possible works inner thread and outer thread.

To acess the page of programming CYCLE70 it's necessary to follow the these steps:

- Press the **[Milling]** softkey
- Press the **[Thread Milling]** softkey
- Insert the data in the blanks
- Press the **[Accept]** softkey



The data to be inputed are the following ones:

RP	Coordinate Z of return of the tool after the end of the cycle (absolute)
SC	Safe distance (distance to approach - without sign)
F1	Feed
4th Blank	Type of machining ▽ = Roughing ▽▽▽ = Finish
5th Blank	Milling direction: Z0 ➔ Z1 : Climb milling Z1 ➔ Z0 : Opposed milling
6th Blank	Milling direction: Righ. = Righ thread Left. = Left thread

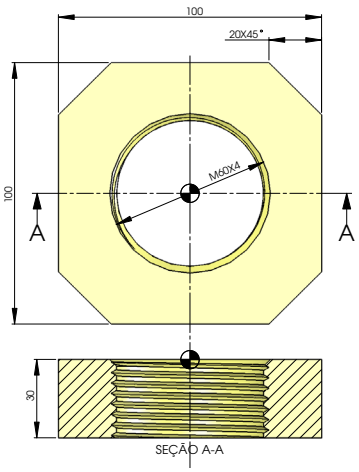
7th Blank	Through the **[SELECT]** key choose the follow options: Inside thread Outside thread
NT	Number of teeth
9th Blank	Through the **[SELECT]** key choose the follow options: **- The individual Positions** (just one hole) - **Holes Model (MCALL)** (it accomplishes several holes with the same depth)
X0	Center point of circle, abscissa
Y0	Center point of circle, ordinate
Z0	Coordinate “Z” of the start point
Z1	Final drilling depth. Choose the option Absolute or Incremental through the **[SELECT]** key.
TABLE	Through the **[SELECT]** key choose the follow options: **ISO métric:** **Withwort:**
D	Core diameter, inside diameter of thread
P	Thread pitch;
Ø	Nominal diameter, outside diameter of thread
H1	Lenght Thread.
DXY	Final machining allowance on “X,Y” axis
U	Final machining allowance on “Z axis”
αS	Initial angle

NOTE:

- *RPM must be programmed in a single block, BEFORE the cycle 70.*
- *Some data can be hidden in the block. This data receive the 0 value.*

Example:

```
WORKPIECE(,,"","CYLINDER",64,0,-30,-80,100)
G17 G71 G90 G94
G53 G0 Z-110 D0
T01;
M6
G54 D01 S1800 M3
G0 X0 Y0
Z2
CYCLE70(5,0,2,-30,60,2.8,0,4,7,1,200,0,0,0,45,11,1,,,,,1,0)
G53 G0 Z-110 D0 M5
M30
```



19.7.1 - Overshoot in the thread longitudinal direction

For thread milling, the travel-in and travel-out movements occur along all three axes concerned. This means that the travel-out movement includes a further step in the vertical axis, beyond the programmed thread depth.

The overshoot is calculated:

$$\triangle z = \frac{p}{4} * \frac{2 * WR + RDIFF}{DIATH}$$

$\triangle z$	-	Overshoot, internal
p	-	Thread pitch
WR	-	Tool radius
DIATH	-	Outside diameter of the thread
RDIFF	-	Radius difference for travel-out For inside threads RDIFF = DIATH/2 - WR, For outside threads RDIFF = DIATH/2 + WR.

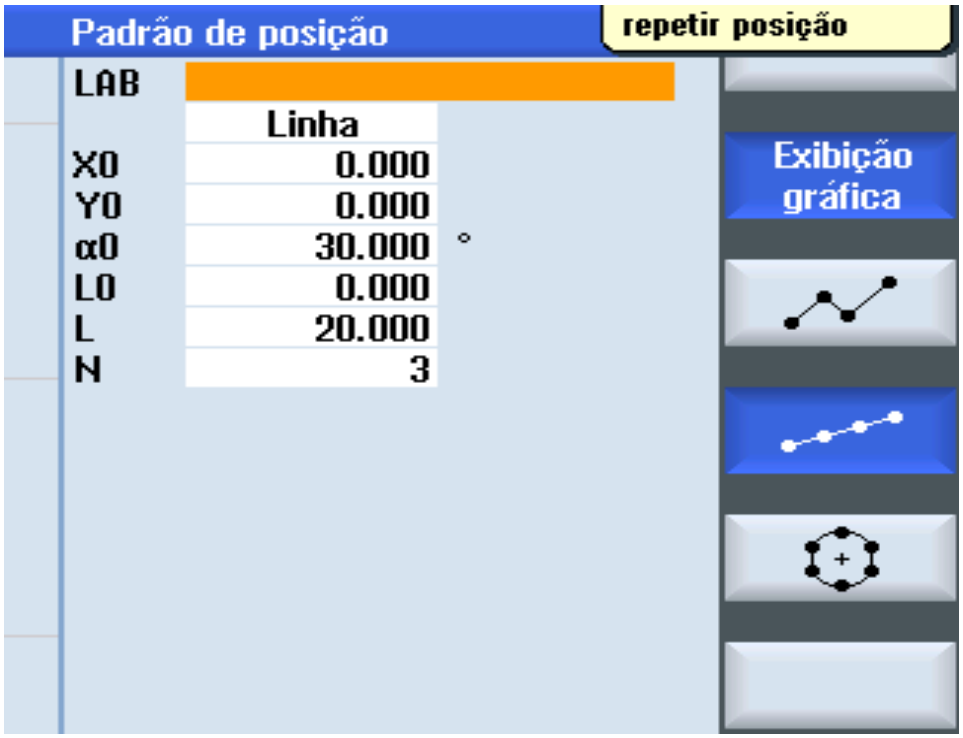
19.8 - HOLES1

Function: Row of holes

With this cycle you can program a row of holes, id est a number of drill roles in a straight line.

To acess the page of programming HOLES 1 it's necessary to follow the these steps:

- Press the **[Drilling]** softkey
- Press the **[Positions]** softkey
- Insert the data in the blanks
- Press the **[Accept]** softkey



The data to be inputed are the following ones:

LAB	Name.
2nd Blank	Through the **[SELECT]** key choose the follow options: - **Line** (HOLES 1) - **Grid** (CYCLE 801) - **Frame** (CYCLE801)
X0	Abscissa of a reference point on the straight line.
Y0	Ordinate of this reference point.
α0	Angle to abscissa.
L0	Distance between the first hole and the reference point.
L	Distance between the holes.
N	Number of holes.

After press the **[Accept]** Softkey, the cycle is inserted in the program according to follow example:

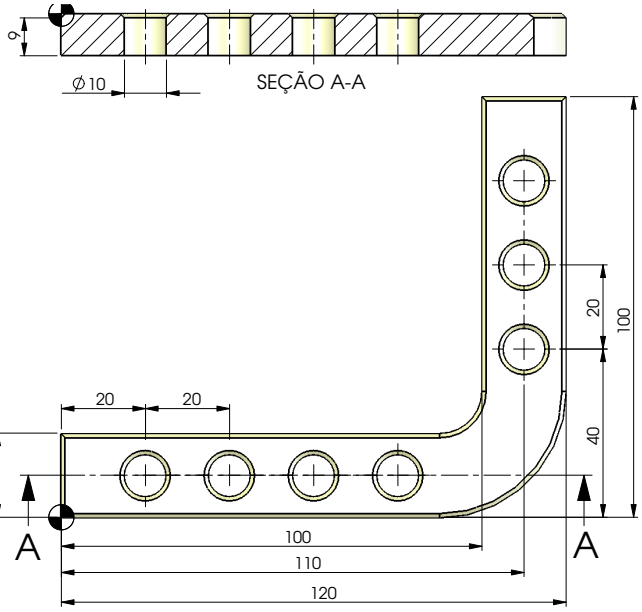
HOLES1 (_____)

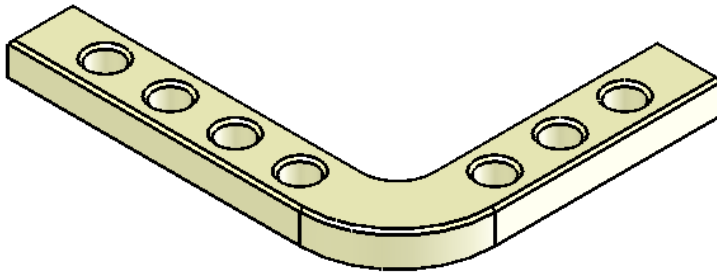
NOTE:

- *Feed and RPM must be programmed in a single block, BEFORE the cycle 83.*
- *It's necessary to program just a value for the end of the drill, that is to say, programming the "Z1" or "E".*
- *Some data can be hidden in the block. This data receive the 0 value.*

Example 1:

```
G17 G71 G90 G94
G53 G0 Z-110 D0
T04
M6
G54 D01 S800 M3
G0 X20 Y10
Z10
F200
MCALL CYCLE82(5,0,2,-26,,0.5,10,1,12)
HOLES1(20,10,0,0,20,4,,0,,,1)
HOLES1(110,40,90,0,20,3,,0,,,1)
MCALL
G53 G0 Z-110 D0 M5
M30
```





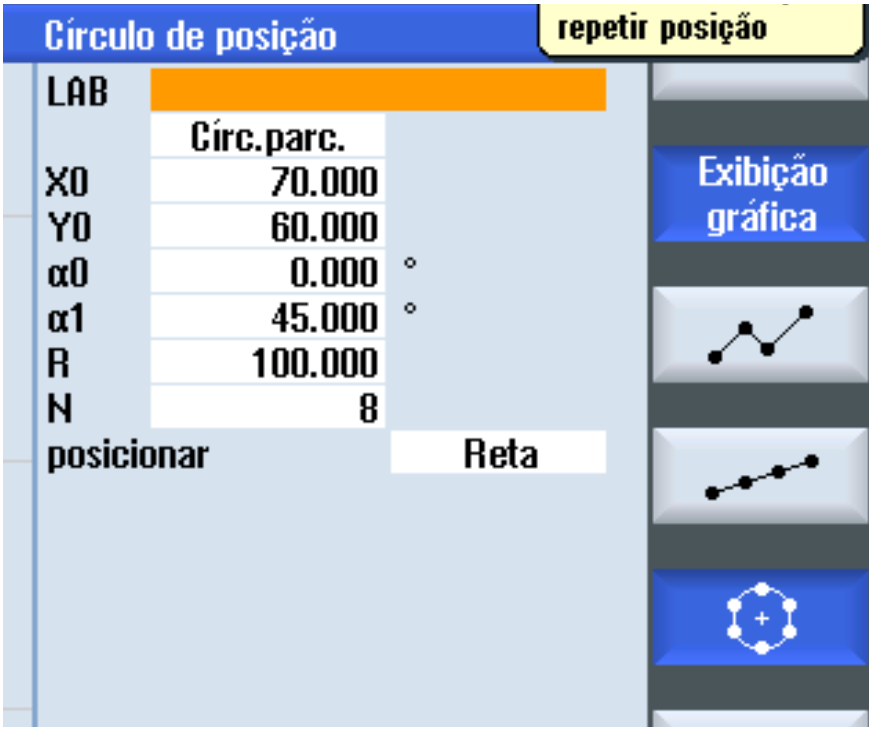
19.9 - HOLES2

Function: Circle of holes

A circle of holes can be machined with this cycle. The machining plan must be defined before the cycle is called. The type of hole is determined by the drilling cycle that has already been called modally.

To acess the page of programming HOLES 2 it's necessary to follow the following steps:

- Press the **[Drilling]** softkey
- Press the **[Positions]** softkey
- Press the **[Circle]** softkey
- Insert the data in the blanks
- Press the **[Accept]** softkey



The data to be inputed are the following ones:

LAB	Name.
2nd Blank	Through the **[SELECT]** key choose the follow options: - **Pitch Circle** - Full **Círcle**
X0	Abscissa of a reference point on the straight line.
Y0	Ordinate of this reference point.
α0	First drill Angle.
α1	Angle between the holes.
R	Radius of circle of holes.

N	Number of holes
Position	Through the **[SELECT]** key choose the follow options: **- Straight** **- Circle**

After press the **[Accept]** Softkey, the cycle is inserted in the program according to follow example:

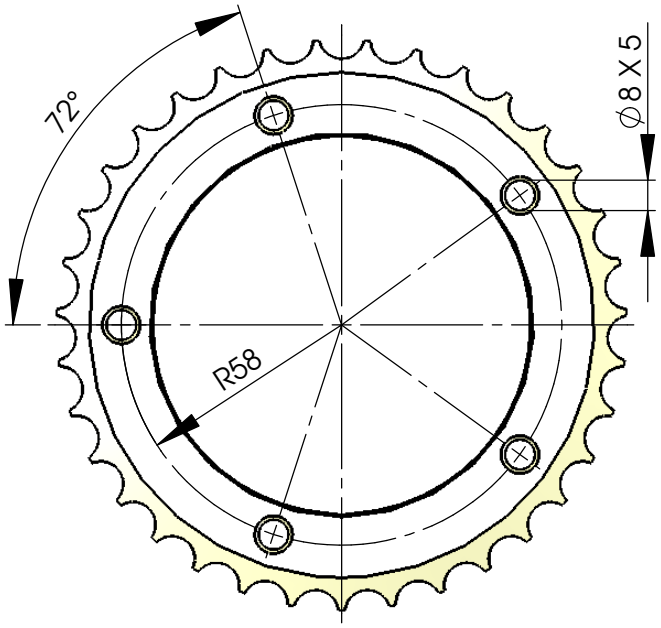
HOLES2 (_____)

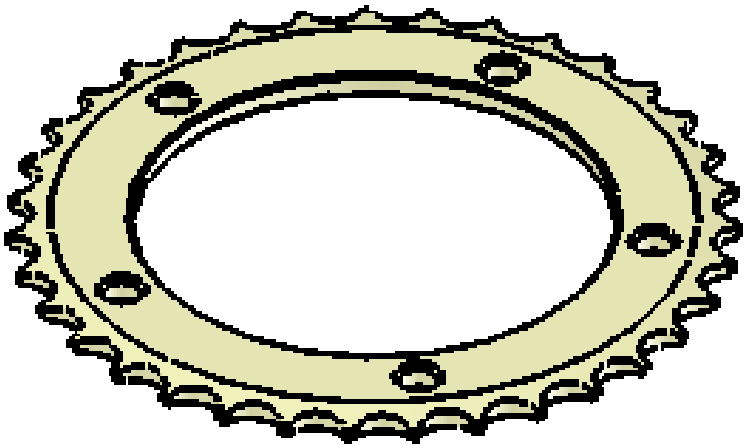
NOTE:

- *The blanks "A" and "B" (center) and the radius define the circle of holes.*
- *Some data can be hidden in the block. This data receive the 0 value.*

Example:

```
N10 WORKPIECE(,,"","CYLINDER",64,0,-
9,-80,135)
G17 G71 G90 G94
G53 G0 Z-110 D0
T08; DRILL D8 MM
M6
G54 D01 G64 CFIN
S1550 M3
G0 X0 Y0
Z10
F230
MCALL CYCLE82(5,0,2,-26,,0.5,10,1,12)
HOLES2(0,0,50,108,30,5,1010,0,,,1)
MCALL
G53 G0 Z-110 D0 M5
M30
```





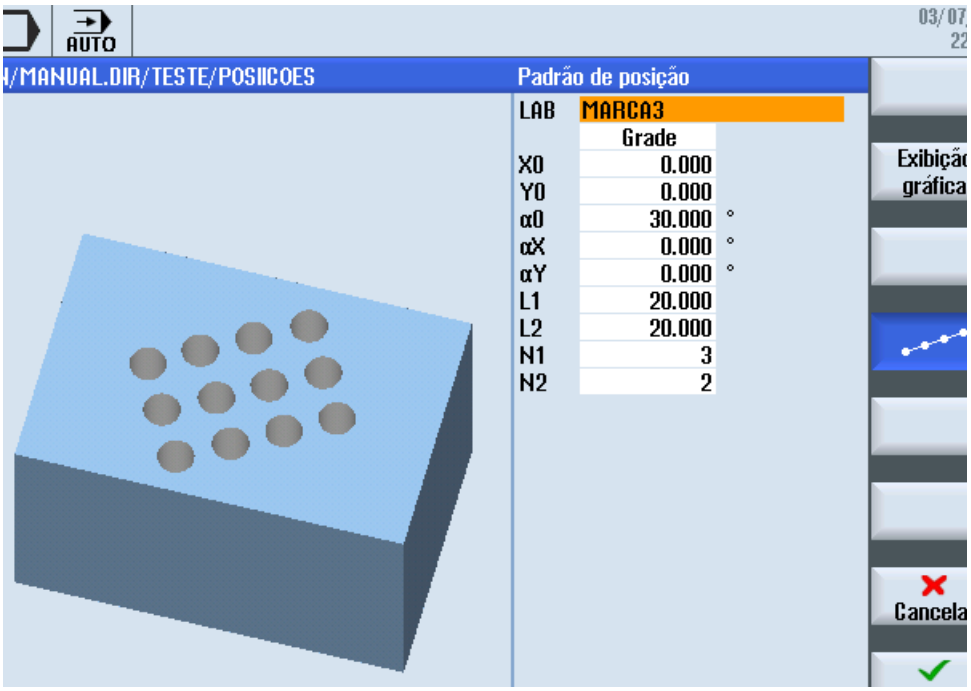
19.10 - CYCLE801

Function : Grid of Holes

Cycle CYCLE801 can be used to machine a “grid of holes”. The type of hole is determined by the drilling cycle that has already been called modally.

To acess the page of programming CYCLE801 it’s necessary to follow these steps:

– Press the **[Drilling]** softkey
– Press the **[Positions]** softkey
– Press the **[Grid]** softkey
– Insert the data in the blanks
– Press the **[Accept]** softkey



The data to be inputed are the following ones:

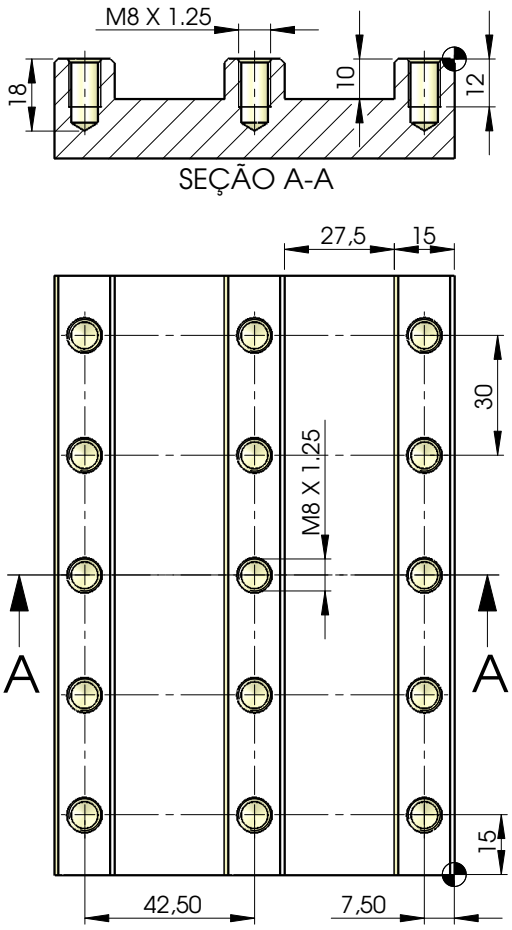
LAB	Name.
2nd Blank	Through the **[SELECT]** key choose the follow options: - **Line** (HOLES 1) - **Grid** (CYCLE 801) - **Frame**(CYCLE801)
X0	Abscissa of a reference point on the straight line.
Y0	Ordinate of this reference point
α0	Angle to abscissa.
L0	Distance between the first hole and the reference point.
L	Distance between the holes.
N	Number of holes.

After press the **[Accept]** Softkey, the cycle is inserted in the program according to follow example:

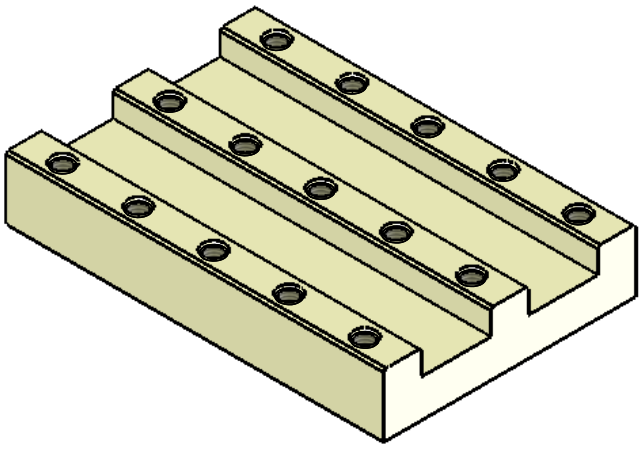
CYCLE801 (_____)

NOTE: *Some data can be hidden in the block. This data receive the 0 value.*

Example 1:




```
WORKPIECE(,,"","BOX",112,0,-25,-80,0,0,-
100,150)
G17 G71 G90 G94
G53 G0 Z0 D0
T01
M6
G54 D01 S1800 M3
G0 X25 Y25
Z10
F100
MCALL CYCLE82(5,0,2,-26,,0.5,10,1,12)
CYCLE801(-92.5,15,0,42.5,30,3,5,0,0,0,0,,,1)
MCALL
G53 G0 Z0 D0
T02
M6
G54 D01 S1800 M3
G0 X25 Y25
Z10
F100
MCALL CYCLE84(5,0,2,-18,,0.7,3,,1.75,5,500,5,0,
1,0,1,5,1,,,,,1001,2001002)
CYCLE801(-92.5,15,0,42.5,30,3,5,0,0,0,0,,,1)
MCALL
G53 G0 Z0 D0
M30
```



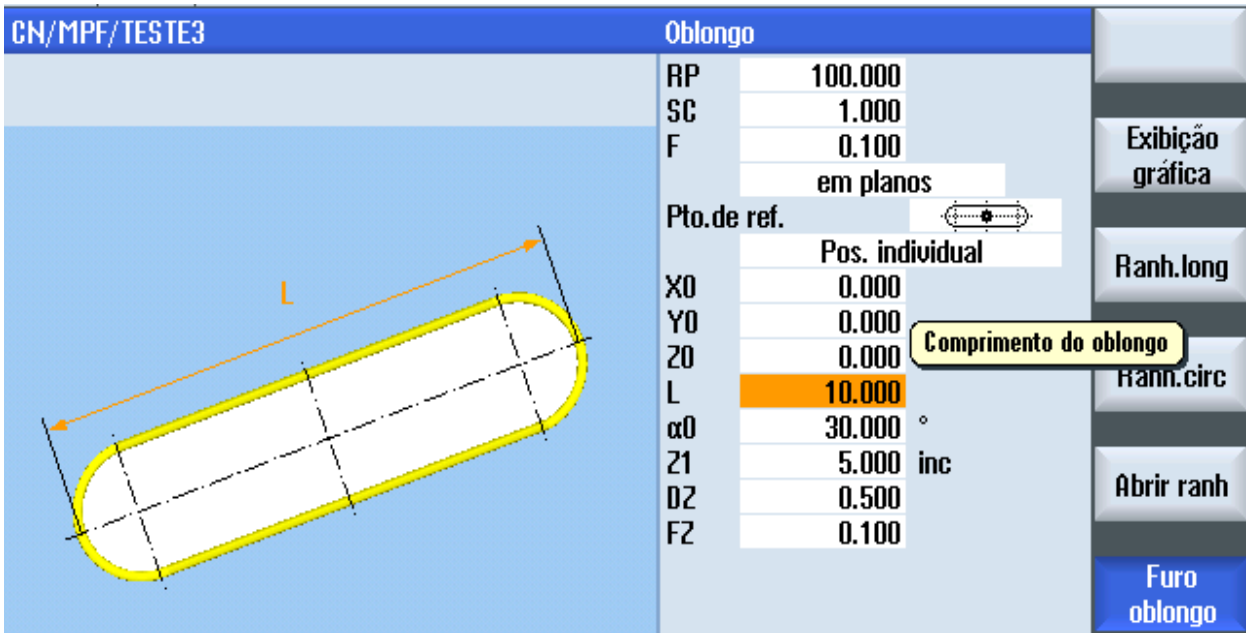
19.11 - LONGHOLE

Function: Longhole

Elongated holes arranged on a circle can be machined with this cycle.

To acess the page of programming of the LONGHOLE it's necessary to follow these steps:

- Press the **[Milling]** softkey
- Press the **[Slot]** softkey
- Press the **[Elongated Hole]** softkey
- Insert the data in the blanks
- Press the **[Accept]** softkey



The data to be inputed are the following ones:

RP	Coordinate Z of return of the tool after the end of the cycle (absolute)
SC	Safe distance (distance to approach - without sign)
F	Feed
4th Blank	Through the **[SELECT]** key choose the follow options: **Centric** **Oscillat**
Reference point	Through the **[SELECT]** key choose the follow options: **Center** **Radius center left** **Radius center right** **Corner left** **Corner Right**
6th Blank	Through the **[SELECT]** key choose the follow options: **- Single Position**(just one hole) - **Position Pattern (MCALL)** (it accomplishes several holes with the same depth)

X0	Center point of circle, abscissa
Y0	Center point of circle, ordinate
Z0	Coordinate “Z” of the start point
L	Length of elongated hole
α0	Initial angle
Z1	Final drilling depth. Choose the option Absolute or Incremental through the **[SELECT]** key.
DZ	Maximum infeed depth
FZ	Feedrate for depth infeed

After press the **[Accept]** Softkey, the cycle is inserted in the program according to follow example:

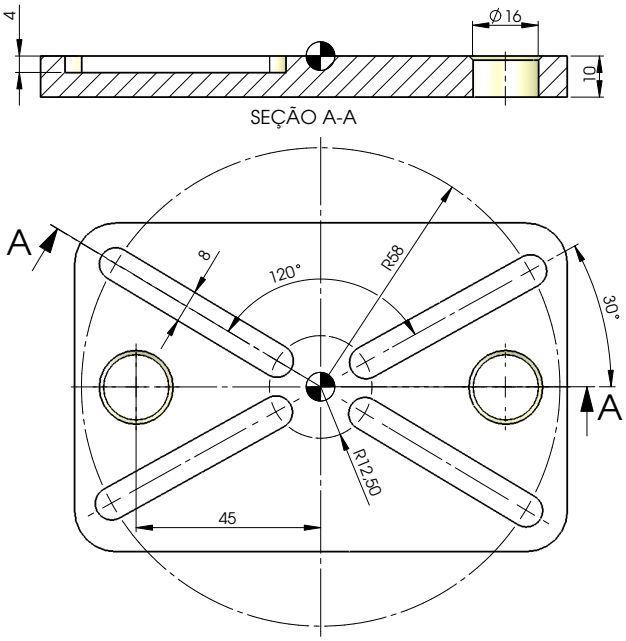
LONGHOLE (_____)

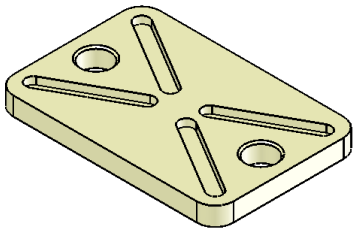
NOTE:

- *A tool offset must be activated before the cycle is called. Otherwise the cycle is aborted and alarm 61000 “No tool offset active” is output.*
- *During the cycle, the workpiece coordinate system is shifted and rotated. The values in the workpiece coordinate system displayed on the actual value display are such that the longitudinal axis of the slot that has just been machined corresponds to the first axis of the current machining plan.*
- *Some data can be hidden in the block. This data receive the 0 value.*

Example:

```
  :
G53 G0 Z-110 D0
T01
M6
G54 D01 S2400 M3 G64 CFIN
G0 X0 Y0
Z10
LONGHOLE(5,0,2,-10,,2,53.5,0,0,8.5,30,12
                0,150,500,2.5,1,0,2100,1001,2)
LONGHOLE(5,0,2,-10, ,2,53.5,0,0, 8.5,210,
                 120,150,500,2.5)
G53 G0 Z-110 D0 M5
M30
```





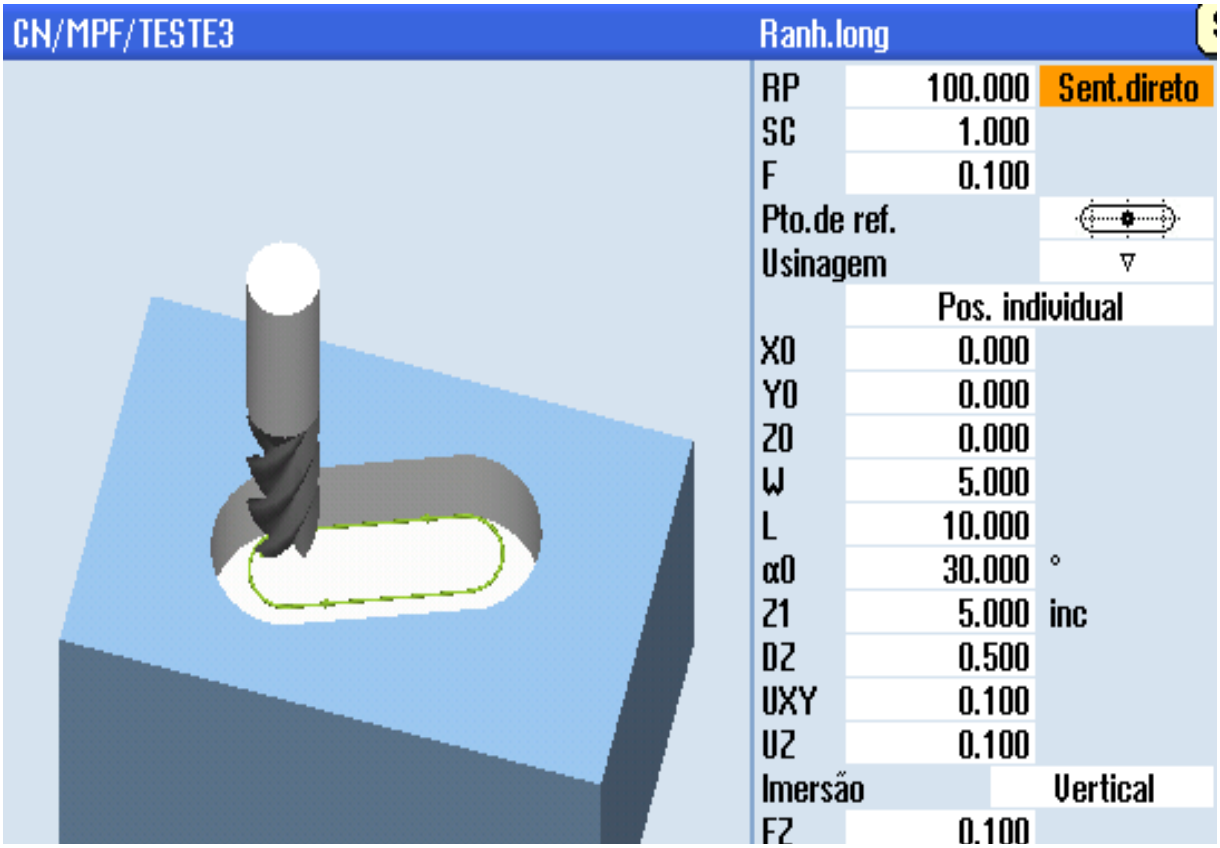
19.12 - SLOT1

Function: Slots arranged on a circle

With this cycle you can machine slots arranged on a circle. The longitudinal axis of the slots is arranged radially.

To acess the page of programming SLOT1 it's necessary to follow these steps:

- Press the **[Milling]** softkey
- Press the **[Slot]** softkey
- Press the **[Longit. Slot]** softkey
- Insert the data in the blanks
- Press the **[Accept]** softkey



The data to be inputed are the following ones:

RP	Coordinate Z of return of the tool after the end of the cycle (absolute)
2nd Blank	Through the **[SELECT]** key choose the follow options: **Up Cut** **Down cut**
SC	Safety distance (distance to approach - without sign)
F	Feed

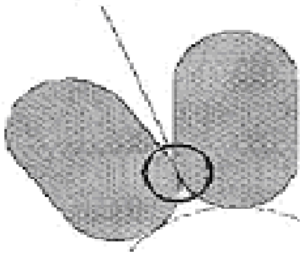
Reference point	Through the **[SELECT]** key choose the follow options: **Center** **Radius center left** **Radius center right** **Corner left** **Corner Right**
Machinning type	Machining type slot ▽ = Roughing ▽▽▽ = Finish
7th Blank	Through the **[SELECT]** key choose the follow options: **- Single Position** (just one hole) - **Position Pattern (MCALL)** (it accomplishes several holes with the same depth)
X0	Center point of circle, abscissa
Y0	Center point of circle, ordinate
Z0	Coordinate “Z” of the start point
W	Length of elongated hole
α0	Initial angle
Z1	Final drilling depth. Choose the option Absolute or Incremental through the **[SELECT]** key.
DZ	Maximum infeed depth
UXY	Final machining allowance on slot edge
UZ	Final machining allowance on the base
Machinning	Through the **[SELECT]** key choose the follow options: **- Vertical** - **Predrilled** - **Oscillat with G1** **- Helical**
FZ	Feedrate for depth infeed

After press the **[Accept]** Softkey, the cycle is inserted in the program according to follow example:

SLOT1 (_____)

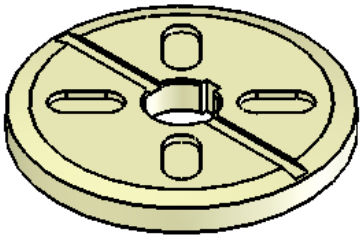
NOTE:

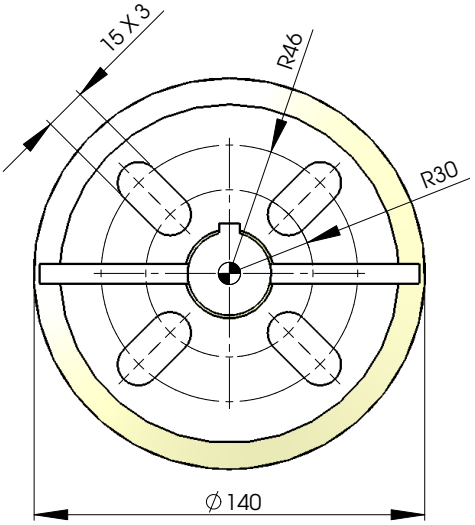
- *A tool offset must be activated before the cycle is called. Otherwise the cycle is aborted and alarm 61000 “No tool offset active” is output.*
- *During the cycle, the workpiece coordinate system is shifted and rotated. The values in the workpiece coordinate system displayed on the actual value display are such that the longitudinal axis of the slot that has just been machined corresponds to the first axis of the current machining plan.*
- *Some data can be hidden in the block. This data receive the 0 value.*
- *If incorrect values are assigned to the parameters that determine the arrangement and size of the slots and thus cause mutual contour violation of the slots, the cycle is not started. The cycle is aborted after the error message 61104 “Contour violation of slots/elongated holes” is output.*



Example:

```
  :
N10 WORKPIECE(,,"","CYLINDER",64,0,-25,-80,140)
N20 G17 G71 G90 G94
N30 ;G53 G0 Z0 D00
N40 T01
N50 M6
N60 G54 D01 S1800 M3
N70 G0 X0 Y0
N80 Z10
N90 F100
MCALL SLOT1(5,0,2,-3,,1,15,6,0,0,5,0,0,0.1,200,1,0,0,21,0.1,15,15,0,2,2,0,1,2,1100,1011,102)
HOLES2(0,0,30,45,30,4,1010,0,,,1)
MCALL
G53 G0 Z-110 D0
M30
```





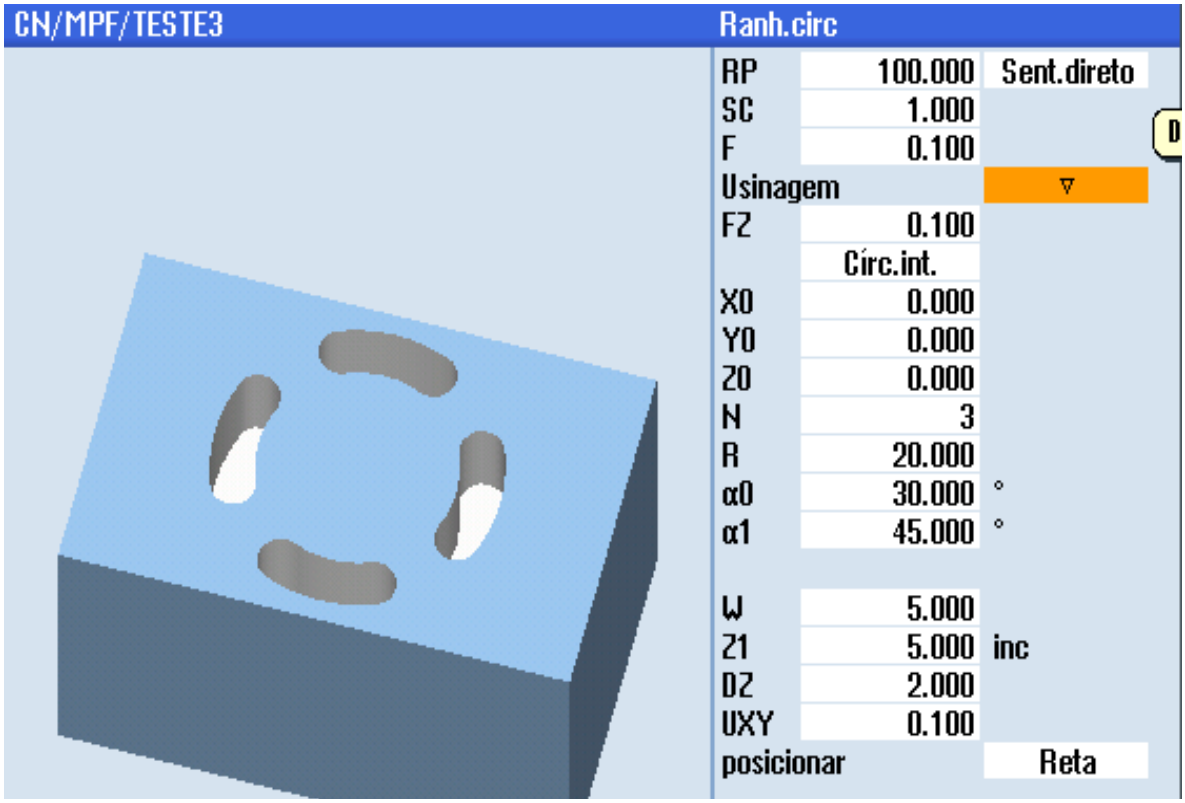
19.13 - SLOT2

Function: Circumferential slots

Cycle SLOT2 is a combined roughing-finishing cycle. With this cycle you can machine circumferential slots arranged on a circle.

To acess the page of programming SLOT1 it's necessary to follow these steps:

- Press the **[Milling]** softkey
- Press the **[Slot]** softkey
- Press the **[Circumfer. Slot]**softkey
- Insert the data in the blanks
- Press the **[Accept]** softkey



The data to be inputed are the following ones:

RP	Coordinate Z of return of the tool after the end of the cycle (absolute)
2nd Blank	Through the **[SELECT]** key choose the follow options: **Up Cut** **Down Cut**
SC	Safety distance (distance to approach - without sign)
F	Feed
Machinning type	Machining type slot ▽ = Roughing ▽▽▽ = Finish

FZ	Feedrate for depth infeed
7th Blank	Through the **[SELECT]** key choose the follow options: - **Círcle incomplete** - **Círcle complete.**
X0	Center point of circle, abscissa
Y0	Center point of circle, ordinate
Z0	Coordinate “Z” of the start point
N	Number of slots
R	Radius of circle
α0	Initial angle
α1	Indexing angle
α2	Angle for the slot length
Z1	Final drilling depth. Choose the option Absolute or Incremental through the **[SELECT]** key.
DZ	Maximum infeed depth
UXY	Final machining allowance on slot edge
UZ	Final machining allowance on the base
Position	Through the **[SELECT]** key choose the follow options: **- Circle** **- Straight**

After press the **[Accept]** Softkey, the cycle is inserted in the program according to follow example:

SLOT2 (_____)

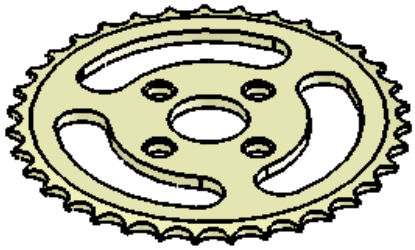
NOTE:

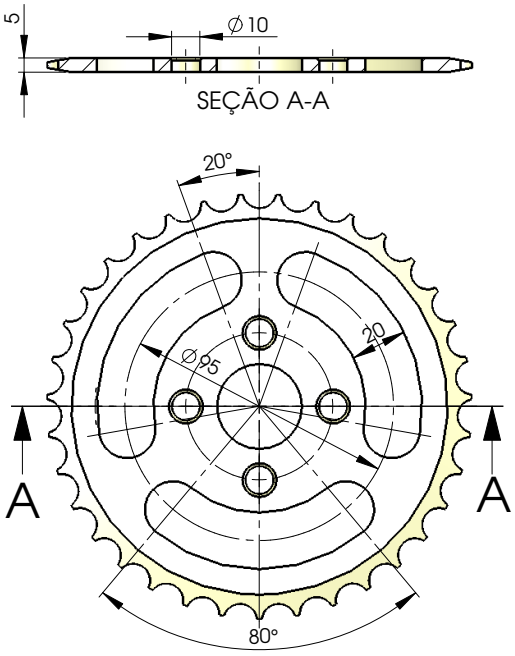
- *During the cycle, the workpiece coordinate system is shifted and rotated. The values in the workpiece coordinate system displayed on the actual value display are such that the longitudinal axis of the slot that has just been machined corresponds to the first axis of the current machining plan.*
- *Some data can be hidden in the block. This data receive the 0 value.*
- *If incorrect values are assigned to the parameters that determine the arrangement and size of the slots and thus cause mutual contour violation of the slots, the cycle is not started.*

Example:

:

```
N10 WORKPIECE(,,"","CYLINDER",64,0,-4.9,-80,140)
N20 G17 G71 G90 G94
N30 G53 G0 Z0 D00
N40 T01
N50 M6
N60 G54 D01 S1800 M3
N70 G0 X0 Y0
N80 Z10
N90 F100
N100 SLOT2(5,0,2,-5,,3,80,20,0,0,47.5,-20,90,200,200,2,0,0,1011,0,0,0,,0,1,2,100,1001,102)
N110 G53 G0 Z-110 D0
N120 M30
```





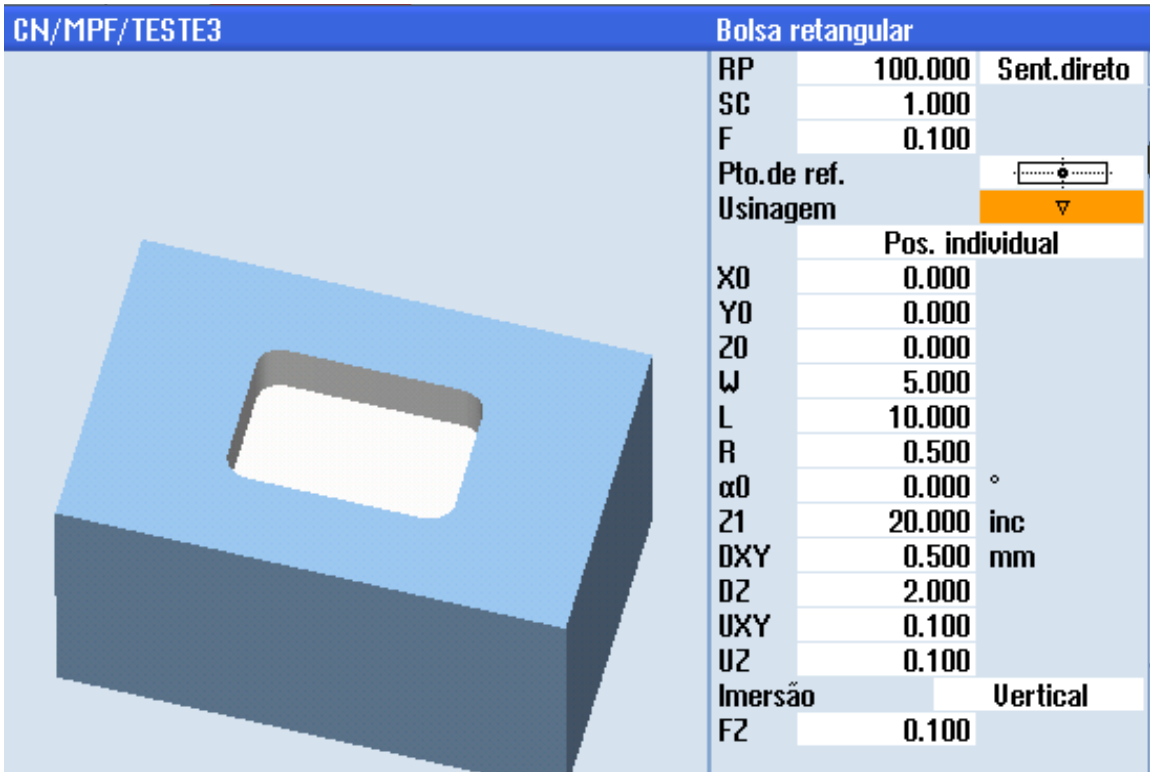
19.14 - POCKET3

Function: Rectangular pockets

The cycle is a combined roughing-finishing cycle. With this cycle you can machine rectangular pockets in any position in the machining plan.

To acess the page of programming POCKET3 it's necessary to follow these steps:

- Press the **[Milling]** softkey
- Press the **[Pocket]** softkey
- Press the **[Rectangular pocket]** softkey
- Insert the data in the blanks
- Press the **[Accept]** softkey



The data to be inputed are the following ones:

RP	Coordinate Z of return of the tool after the end of the cycle (absolute)
2nd Blank	Through the **[SELECT]** key choose the follow options: **Up Cut** **Down Cut**
SC	Safety distance (distance to approach - without sign)
F	Feed
Machinning type	Machining type slot ▽ = Roughing ▽▽▽ = Finish

6th Blank	Through the **[SELECT]** key choose the follow options: - **Single Position** - **Position Pattern** (MCALL)**.**
X0	Center point of rectangle, abscissa
Y0	Center point of rectangle, ordinate
Z0	Coordinate “Z” of the start point
W	Pocket length
L	Pocket width
R	Radius of circle
α0	Initial angle
Z1	Final drilling depth. Choose the option Absolute or Incremental through the **[SELECT]** key.
DZ	Maximum infeed depth
UXY	Final machining allowance on slot edge
UZ	Final machining allowance on the base
Machinning	Through the **[SELECT]** key choose the follow options: **- Vertical** - **Predrilled** - **Oscillat G1** **- Helical**
FZ	Feedrate for depth infeed

After press the **[Accept]** Softkey, the cycle is inserted in the program according to follow example:

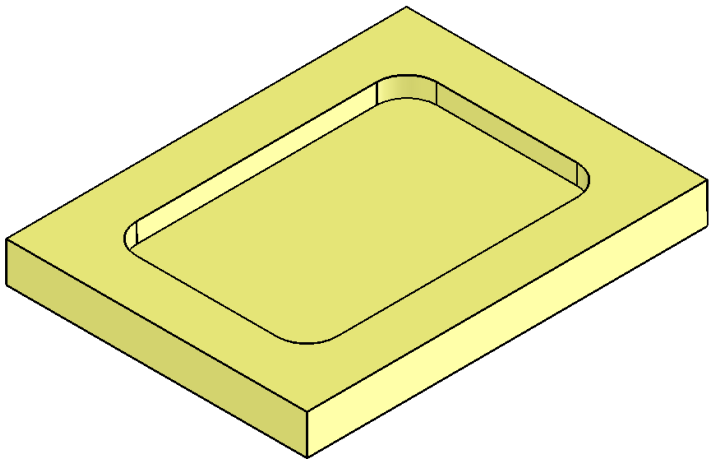
POCKET3 (_____)

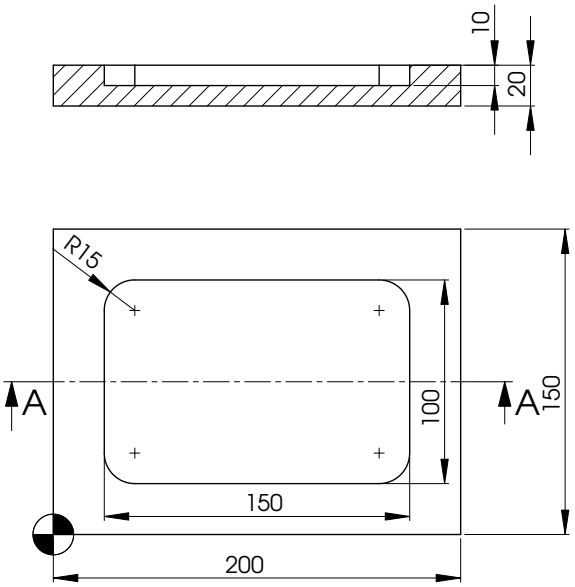
NOTE:

- *The starting position can be any position from which each of the elongated holes can be approached without collision.*
- *A tool offset must be activated before the cycle is called. Otherwise the cycle is aborted and alarm 61000 “No tool offset active” is output.*
- *Some data can be hidden in the block. This data receive the 0 value.*
- *It’s necessary insert the “0” value in the blank MID to remove all the material in a only one depth .*

Example:

```
:
WORKPIECE(,,"","BOX",112,0,-20,-80,0,0,200,150)
G17 G71 G90 G94
G53 G0 Z0 D00
T01
M6
G54 D01 S1800 M3
G0 X0 Y0
Z10
F100
POCKET3(5,0,2,-10,150,100,15,100,75,0,2,0.1,0.1,200,0.1,0,21,50,8,3,15,10,1,0,1,2,1110
0,11,110)
POCKET3(5,0,2,-10,150,100,15,100,75,0,2,0.1,0.1,200,0.1,0,22,50,8,3,15,10,1,0,1,2,1110
0,11,110)
G53 G0 Z0 D0
M30
```





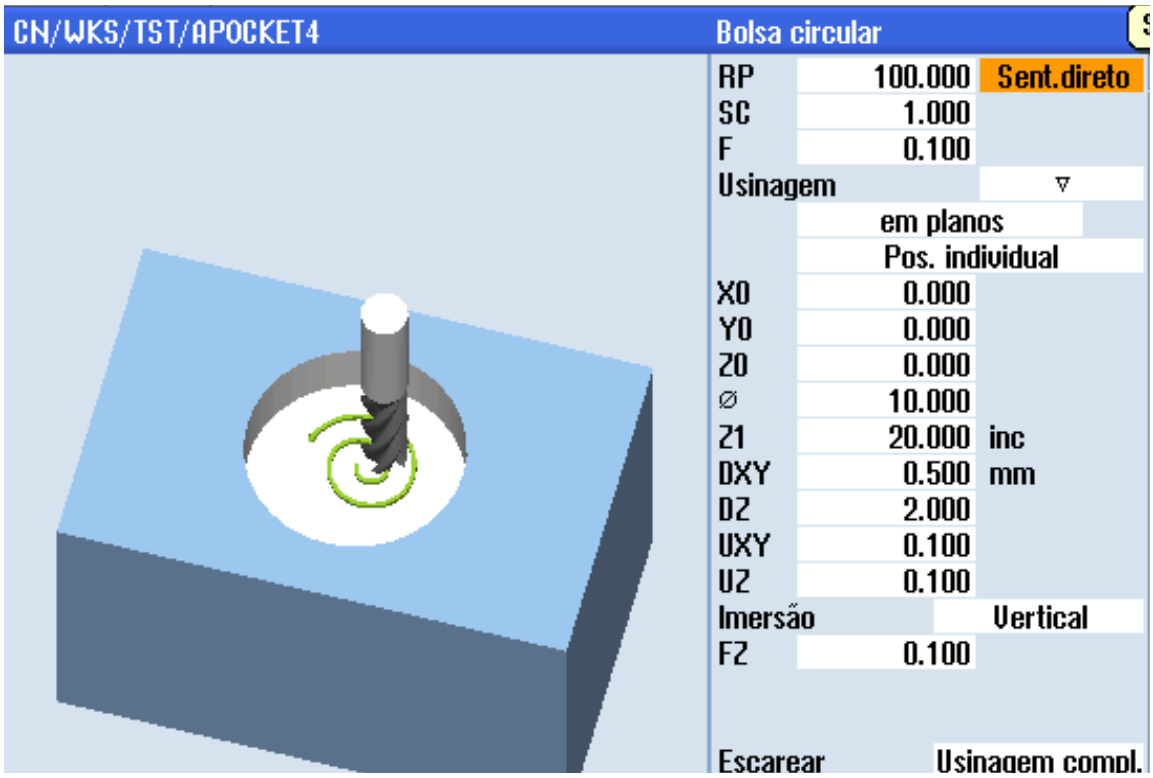
19.15 - POCKET4

Function: Circular pockets

The cycle is a combined roughing-finishing cycle. With this cycle you can machine circular pockets in the machining plan.

To acess the page of programming POCKET 4 it's necessary to follow these steps:

- Press the **[Milling]** softkey
- Press the **[Pocket]** softkey
- Press the **[Circular Pocket]** softkey
- Insert the data in the blanks
- Press the **[Accept]** softkey



The data to be inputed are the following ones:

RP	Coordinate Z of return of the tool after the end of the cycle (absolute)
2nd Blank	Through the **[SELECT]** key choose the follow options: **Up Cut** **Down Cut**
SC	Safety distance (distance to approach - without sign)
F	Feed
Machinning type	Machining type slot ▽ = Roughing ▽▽▽ = Finish

6th Blank	Through the **[SELECT]** key choose the follow options: - **Single Position** - **Position Pattern**
X0	Center point of rectangle, abscissa
Y0	Center point of rectangle, ordinate
Z0	Coordinate “Z” of the start point
W	Pocket length
L	Pocket width
R	Radius of circle
α0	Initial angle
Z1	Final drilling depth. Choose the option Absolute or Incremental through the **[SELECT]** key.
DZ	Maximum infeed depth
UXY	Final machining allowance on slot edge
UZ	Final machining allowance on the base
Machinning	Through the **[SELECT]** key choose the follow options: **- Vertical** - **Predrilled** - **Oscillat with G1** **- Helical**
FZ	Feedrate for depth infeed
Position	Through the **[SELECT]** key choose the follow options: **- Circle** **- Straight**

After press the **[Accept]** Softkey, the cycle is inserted in the program according to follow example:

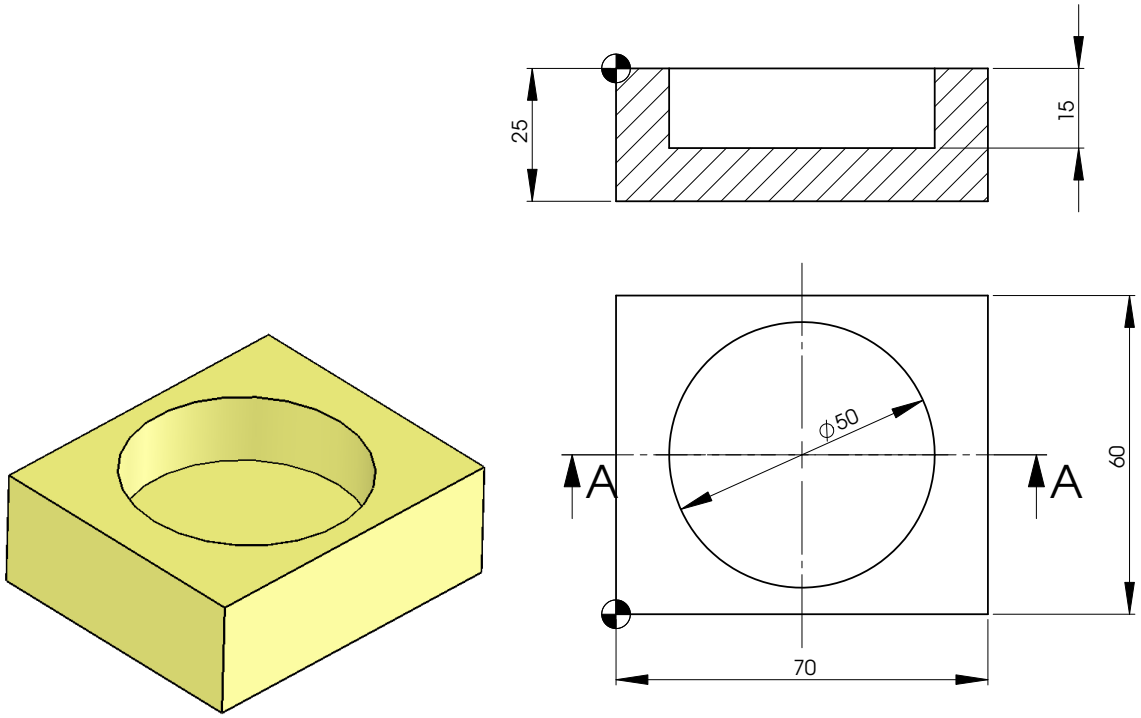
POCKET3 (_____)

NOTE:

- *A tool offset must be activated before the cycle is called. Otherwise the cycle is aborted and alarm 61000 “No tool offset active” is output.*
- *Some data can be hidden in the block. This data receive the 0 value.*
- *It’s necessary insert the “0” value in the blank MID to remove all the material in a only one depth .*

Example:

```
WORKPIECE(,,"","BOX",112,0,-20,-80,0,0,70,60)
G53 G0 Z0 D0
T01
M6
G54 D01 S1800 M3
G0 X0 Y0
Z10
F100
POCKET3(5,0,2,-10,150,100,15,100,75,0,2,0.1,0.1,200,0.1,0,21,50,8,3,15,10,1,0,1,2,1110
0,11,110)
POCKET4(5,0,2,-15,50,35,30,2,0.1,0.1,200,0.1,0,1011,60,9,15,0,2,0,1,2,10100,111,110)
POCKET4(5,0,2,-15,50,35,30,2,0.1,0.1,200,0.1,0,1012,60,9,15,0,2,0,1,2,10100,111,110)
M30
```



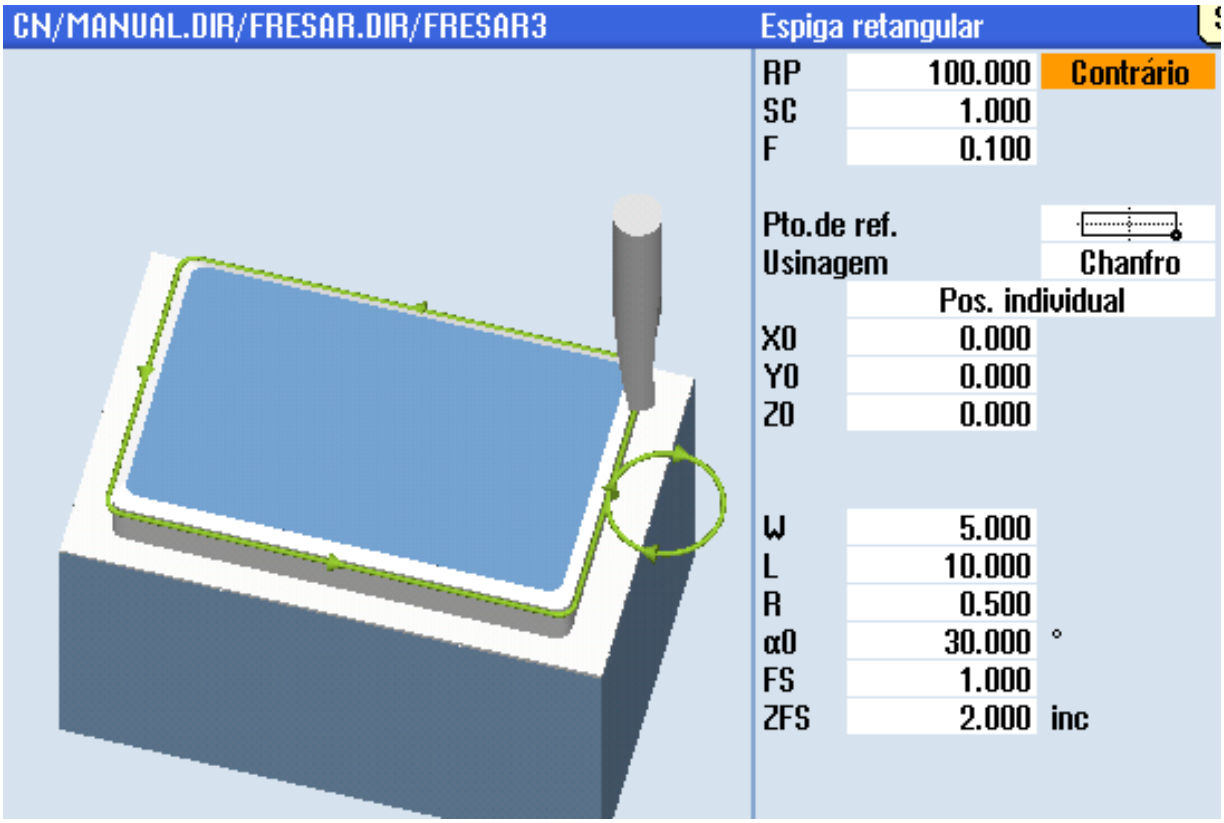
19.16 - CYCLE76

Function: Rectangular spigots

With this cycle you can machine rectangular spigots in the machining plan.

To acess the page of programming CYCLE 76 it's necessary to follow these steps:

- Press the **[Milling]** softkey
- Press the **[Multi Edge Spigots]** softkey
- Press the **[Rectangular spigots]** softkey
- Insert the data in the blanks
- Press the **[Accept]** softkey



The data to be inputed are the following ones:

RP	Coordinate Z of return of the tool after the end of the cycle (absolute)
2nd Blank	Through the **[SELECT]** key choose the follow options: **Up Cut** **Down Cut**
SC	Safety distance (distance to approach - without sign)
F	Feed

Machinning type	Machining type slot ▽ = Roughing ▽▽▽ = Finish
6th Blank	Through the **[SELECT]** key choose the follow options: - **Single position** - **Position Pattern** (MCALL)
X0	Center point of rectangle, abscissa
Y0	Center point of rectangle, ordinate
Z0	Coordinate “Z” of the start point
W1	Length of blank spigot
L1	Width of blank spigot
W	Pocket length
L	Pocket width
R	Radius of circle
α0	Initial angle
Z1	Final drilling depth. Choose the option Absolute or Incremental through the **[SELECT]** key.
DZ	Maximum infeed depth
UXY	Final machining allowance on slot edge
UZ	Final machining allowance on the base

After press the **[Accept]** Softkey, the cycle is inserted in the program according to follow example:

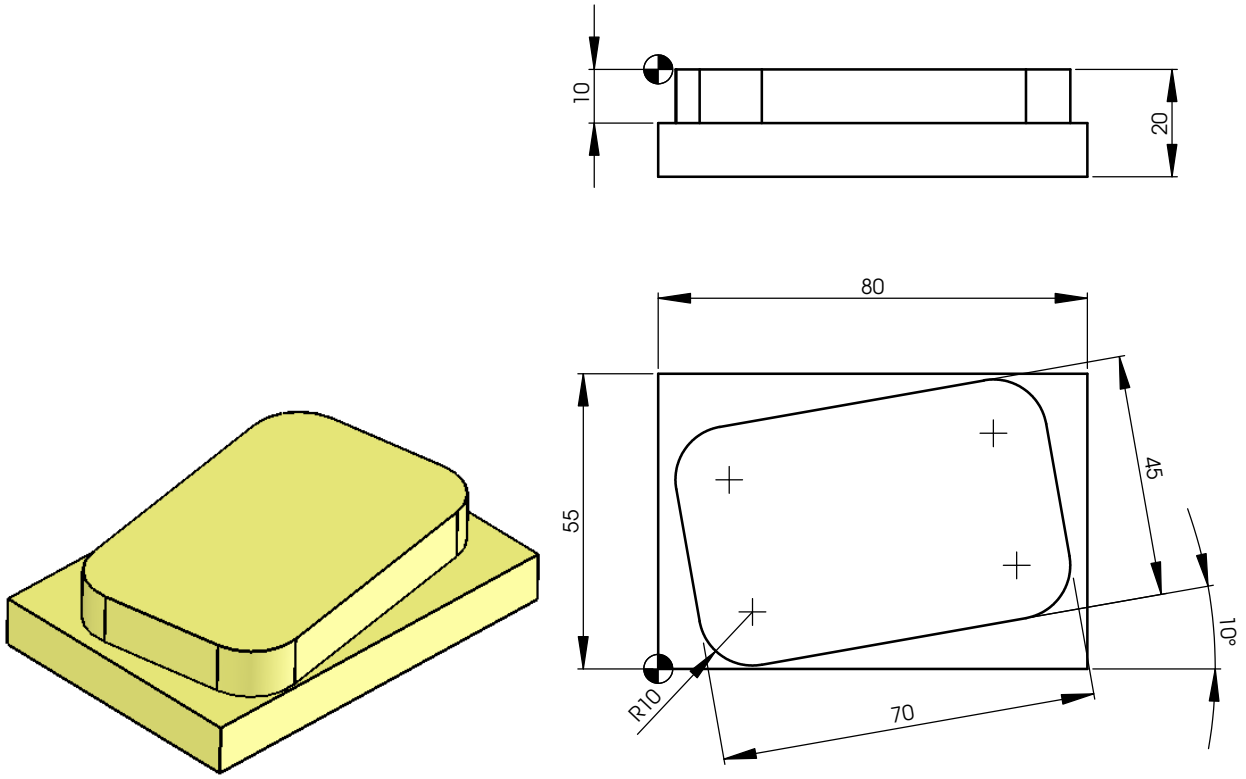
CYCLE 76 (_____)

NOTE:

- *A tool offset must be activated before the cycle is called. Otherwise the cycle is aborted and alarm 61000 “No tool offset active” is output.*
- *Some data can be hidden in the block. This data receive the 0 value.*

Example:

```
WORKPIECE(,,"","BOX",112,0,-25,-80,0,0,80,55)
G17 G71 G90 G94
G53 G0 Z0 D0
T01
M6
G54 D01 S1800 M3
G0 X25 Y25
Z10
F100
CYCLE76(5,0,2,-10,,70,45,10,40,27.5,10,2,0.1,0.1,200,150,0,1,80,55,1,2,1100,1,102)
CYCLE76(5,0,2,-10,,70,45,10,40,27.5,10,2,0.1,0.1,200,150,0,1,80,55,1,2,1100,1,102)
G53 G0 Z0 D0
M30
```



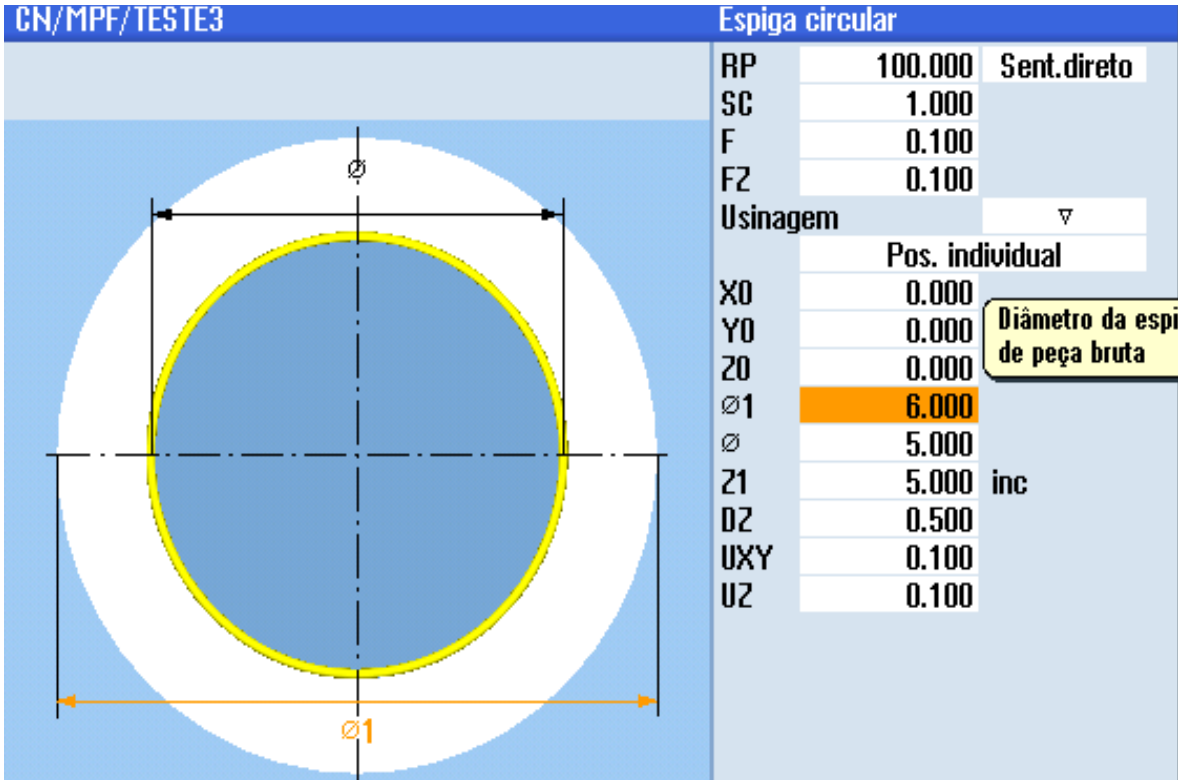
19.17 - CYCLE77

Aplicação: Circular spigots

With this cycle you can machine circular spigots in the machining plan. For finishing, a face cutter is needed.

To acess the page of programming CYCLE 77 it's necessary to follow these steps:

- Press the **[Milling]** softkey
- Press the **[Multi Edge Spigots]** softkey
- Press the **[Circular spigots]** softkey
- Insert the data in the blanks
- Press the **[Accept]** softkey



The data to be inputed are the following ones:

RP	Coordinate Z of return of the tool after the end of the cycle (absolute)
2nd Blank	Through the **[SELECT]** key choose the follow options: **Up Cut** **Down Cut**
SC	Safety distance (distance to approach - without sign)
F	Feedrate on contour
FZ	Feedrate for depth infeed

Machinning type	Machining type slot ▽ = Roughing ▽▽▽ = Finish
7th Blank	Through the **[SELECT]** key choose the follow options: - **Single Position** - **Position Pattern** (MCALL)
X0	Center point of circle, abscissa
Y0	Center point of circle, ordinate
Z0	Coordinate “Z” of the start point
Ø1	Diameter of spigot
Ø	Diameter of blank
Z1	Final drilling depth. Choose the option Absolute or Incremental through the **[SELECT]** key.
DZ	Maximum infeed depth
UXY	Final machining allowance on slot edge
UZ	Final machining allowance on the base of slot

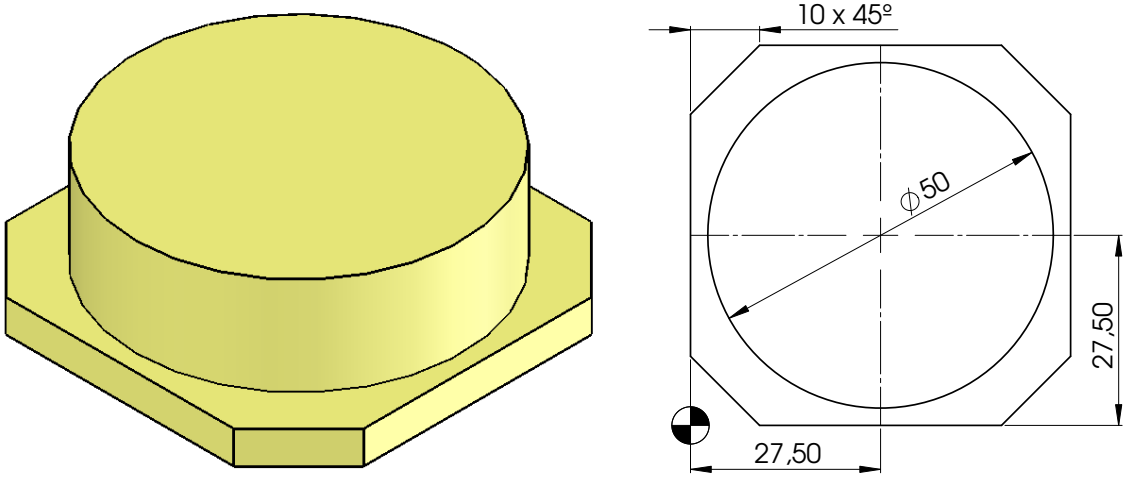
NOTE:

– *A tool offset must be activated before the cycle is called. Otherwise the cycle is aborted and alarm 61000 “No tool offset active” is output.*

– *Some data can be hidden in the block. This data receive the 0 value.*

Example:

```
WORKPIECE(,,"","BOX",112,0,-20,-80,0,0,55,55)
G53 G0 Z0 D0
T01
M6
G54 D01 S1800 M3
G0 X25 Y25
Z10
F100
CYCLE76(5,0,2,-10,,70,45,10,40,27.5,10,2,0.1,0.1,200,150,0,1,80,55,1,2,1100,1,102)
CYCLE77(5,0,2,-15,,50,27.5,27.5,2,0.1,0.1,200,150,0,1,55,1,2,100,1,102)
CYCLE77(5,0,2,-15,,50,27.5,27.5,2,0.1,0.1,200,150,0,2,55,1,2,100,1,102)
M30
```



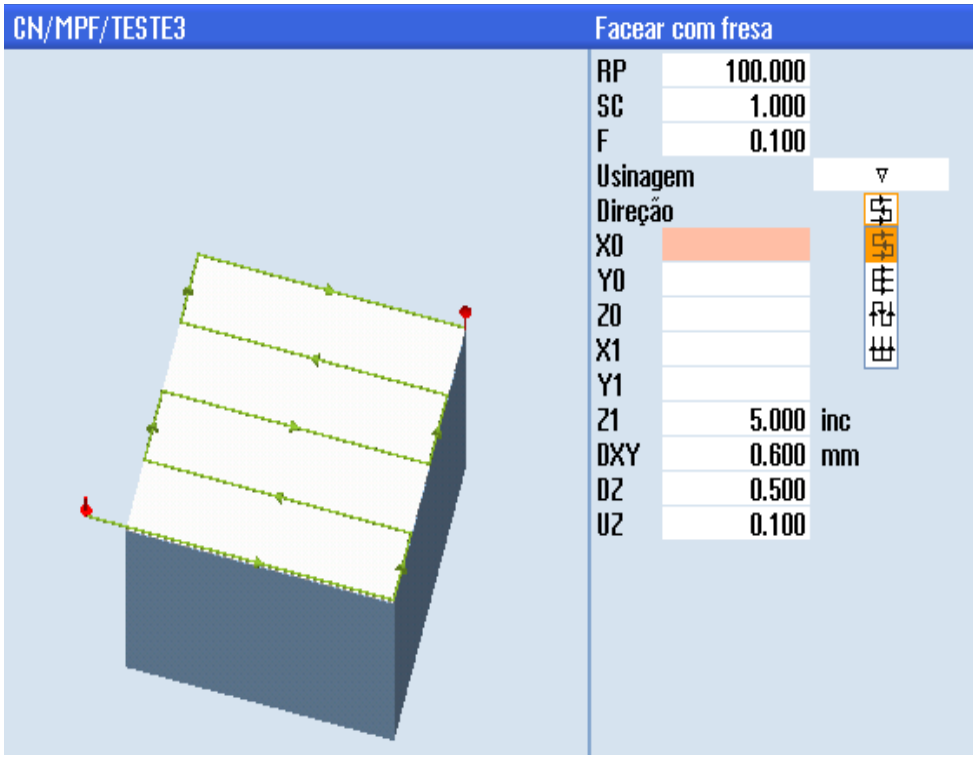
19.18 - CYCLE61

Function: Face milling

With cycle CYCLE61, you can face mill any rectangular surface.

To acess the page of programming CYCLE 61 it's necessary to follow these steps:

- Press the **[Milling]** softkey
- Press the **[Face milling]** softkey
- Insert the data in the blanks
- Press the **[Accept]** softkey



The data to be inputed are the following ones:

RP	Coordinate Z of return of the tool after the end of the cycle (absolute)
SC	Safety distance (distance to approach - without sign)
F	Feed
Machinning type	Machining type slot ▽ = Roughing ▽▽▽ = Finish
Direction	Through the **[SELECT]** key choose the follow options: - **Same Machining Direction** - **Changing Machining Direction**
X0	Starting point, abscissa

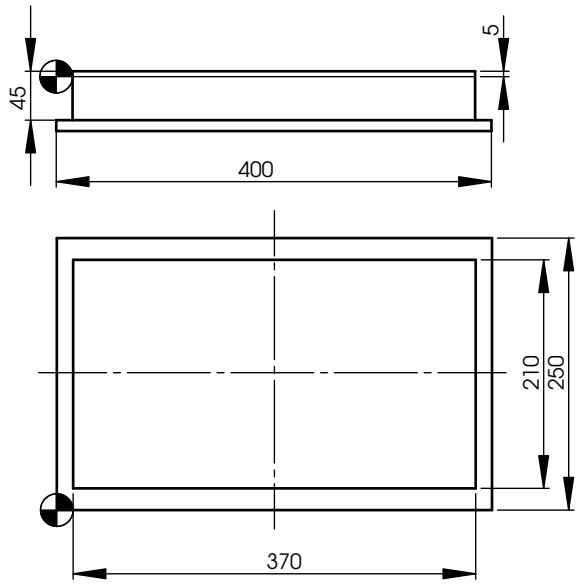
Y0	Starting point, ordinate
Z0	Coordinate “Z” of the start point
X1	Finishing point , abscissa
Y1	Finishing point , ordinate
Z1	Depth
UXY	Final machining allowance on slot edge
UZ	Final machining allowance on the base

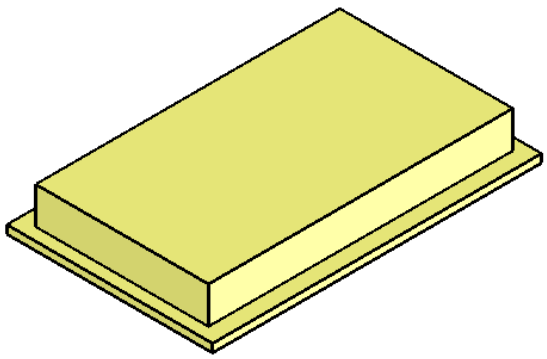
NOTE:

- *A tool offset must be activated before the cycle is called. Otherwise the cycle is aborted and alarm 61000 “No tool offset active” is output.*
- *Some data can be hidden in the block. This data receive the 0 value.*

Example:

```
WORKPIECE(,,"","BOX",112,5,-20,-80,0,0,370,250)
G53 G0 Z0 D0
T01
M6
G54 D01 S1800 M3
G0 X25 Y25
Z10
F100
CYCLE61(8,5,2,0,0,0,370,250,2,80,0.1,200,31,0,1,
11010)
CYCLE61(8,5,2,0,0,0,370,250,2,80,0,200,32,0,1,11
010)
G53 G0 Z0 D0
M30
```





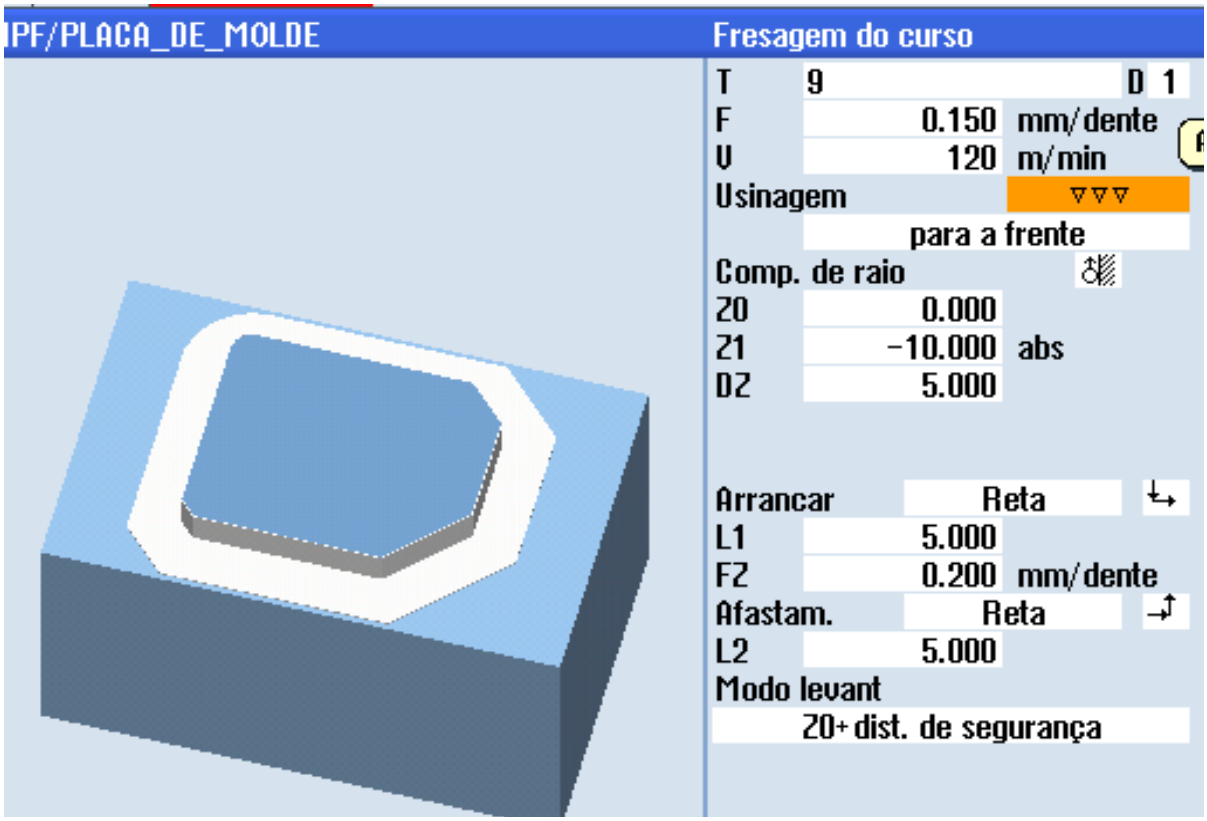
19.19 - CYCLE72

Function: Path milling

With the cycle CYCLE72 it is possible to mill along any contour defined in a subroutine.

To acess the page of programming CYCLE 72 it's necessary to follow these steps:

- Press the **[Cont. Mill.]** softkey
- Press the **[Path Milling]** softkey
- Insert the data in the blanks
- Press the **[Accept]** softkey



The data to be inputed are the following ones:

RP	Coordinate Z of return of the tool after the end of the cycle (absolute)
SC	Safety distance (distance to approach - without sign)
F	Feedrate on contour
Machinning type	Machining type slot ▽ = Roughing ▽▽▽ = Finish
5th Blank	Through the **[SELECT]** key choose the follow options: - **Forward** - **BackForward**

Radius Comp	- **Off** - **Left of the Contour** **- Right of the Contour**
Z0	Coordinate “Z” of the start point
Z1	Final drilling depth. Choose the option Absolute or Incremental through the **[SELECT]** key.
DZ	Maximum infeed depth
UXY	Final machining allowance on slot edge
UZ	Final machining allowance on the base
Approach	- **Straight** **- Quarter Circle** **- Semi Circle** **- vertical**
FZ	Infeed feedrate
Retract	- **Straight** **- Quarter Circle** **- Semi Circle** **- vertical**
Last Blank	Through the **[SELECT]** key choose the follow options: **0 Return at end of contour to _RP** **1 Return at end of contour to _Z0 + _SC** **2 Return at end of contour to _SC** **3 No return to end of contour**

NOTE:

– *A tool offset must be activated before the cycle is called. Otherwise the cycle is aborted and alarm 61000 “No tool offset active” is output.*

– *Some data can be hidden in the block. This data receive the 0 value.*

20 - FRAME FUNCTIONS

20.1 - FUNCTIONS: TRANS, ATRANS

Programmable zero offset

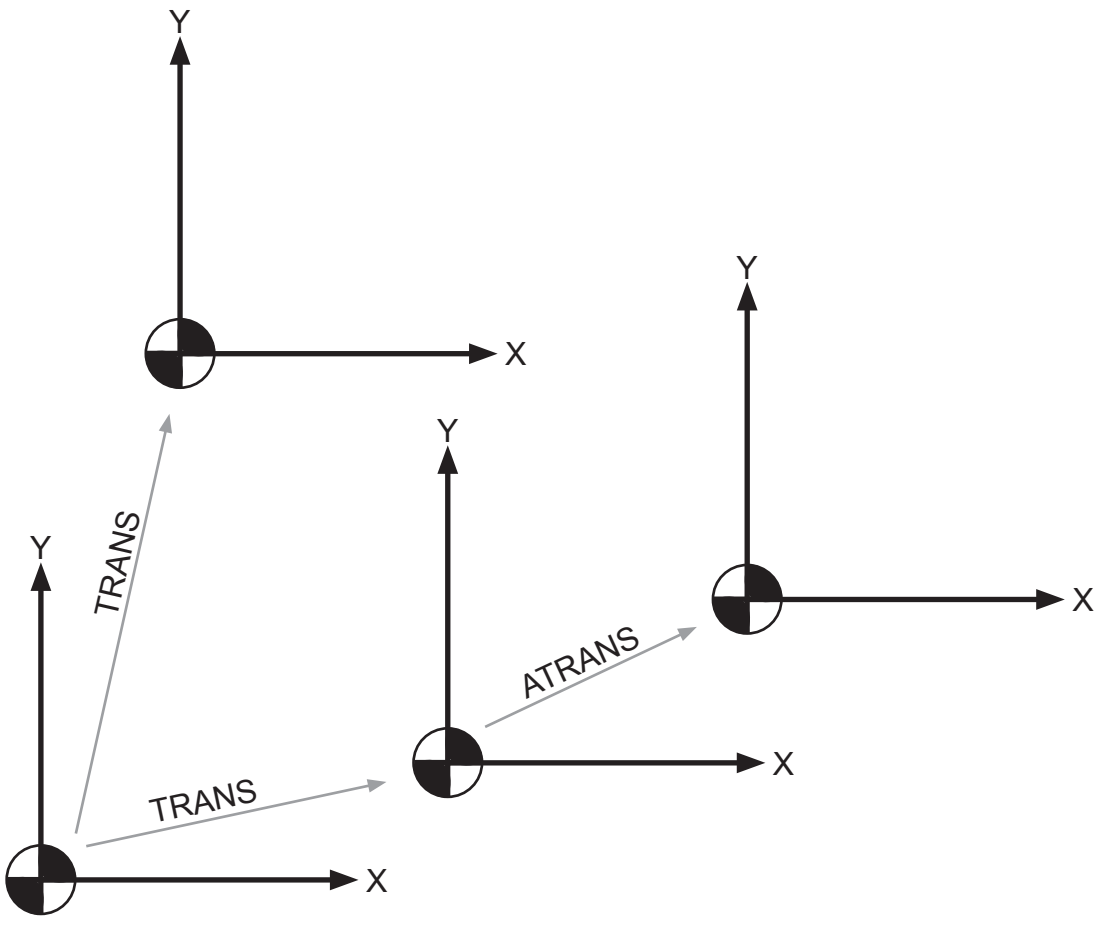
TRANS/ATRANS can be used to program translations for all path and positioning axes in the direction of the specified axis. This allows you to work with different zero points, for example when performing recurring machining processes at different workpiece positions.

TRANS X Y Z - Translation through the offset values programmed in the specified axis directions (path, synchronized axes and positioning axes). The last specified settable zero offset (G54 to G599) is used as a reference.

ATRANS X Y Z - Translation through the offset values programmed in the specified axis directions. The currently set or last programmed zero point is used as the reference.

Deactivate programmable zero offset For all axes: TRANS (without axis parameter)

Syntax:
TRANS X___ Y___ Z___



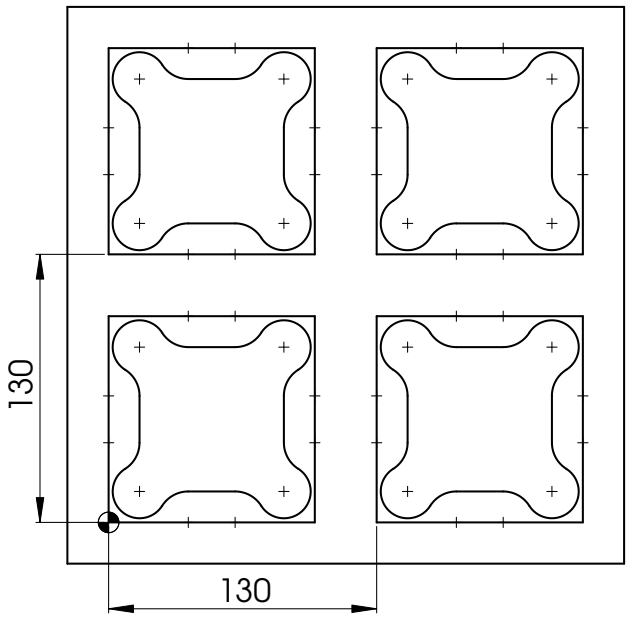
Example:

G17 G71 G90 G94
G53 G0 Z-110 D0
T01; MILL D20
M6
G54 D01 G64 CFIN
S2000 M3
PATH
TRANS X130
PATH
TRANS Y130
PATH
TRANS X130 Y130
;or ATRANS X130
PATH
TRANS
G53 G0 Z-110 D0 M5
M30

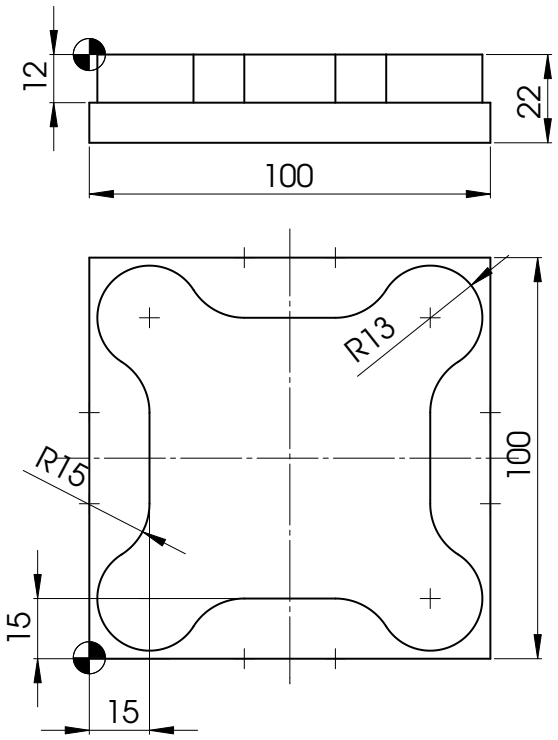
SUB PROGRAM:

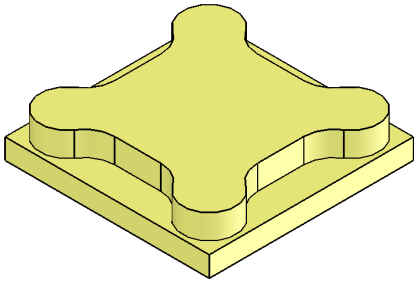
PATH.SPF
G0 X50 Y-15
Z0
STA: G1 Z=IC(-4) F800
G41 Y15
X28 RNDM=15
G2 X15 Y28 CR=-13
G1 Y72
G2 X28 Y85 CR=-13
G1 X72
G2 X85 Y72 CR=-13
G1 Y28
G2 X72 Y15 CR=-13
G1 X50 RNDM=0
END: G40 Y-15
REPEAT STA END P2
G0 Z5
M17

DEVICE:



PEÇA:





20.2 - FUNCTION: ROT, AROT

Programmable rotation

ROT/AROT can be used to rotate the workpiece coordinate system around each of the geometry axes X, Y, Z or through an angle RPL in the selected working plan G17 to G19 (or around the perpendicular infeed axis).

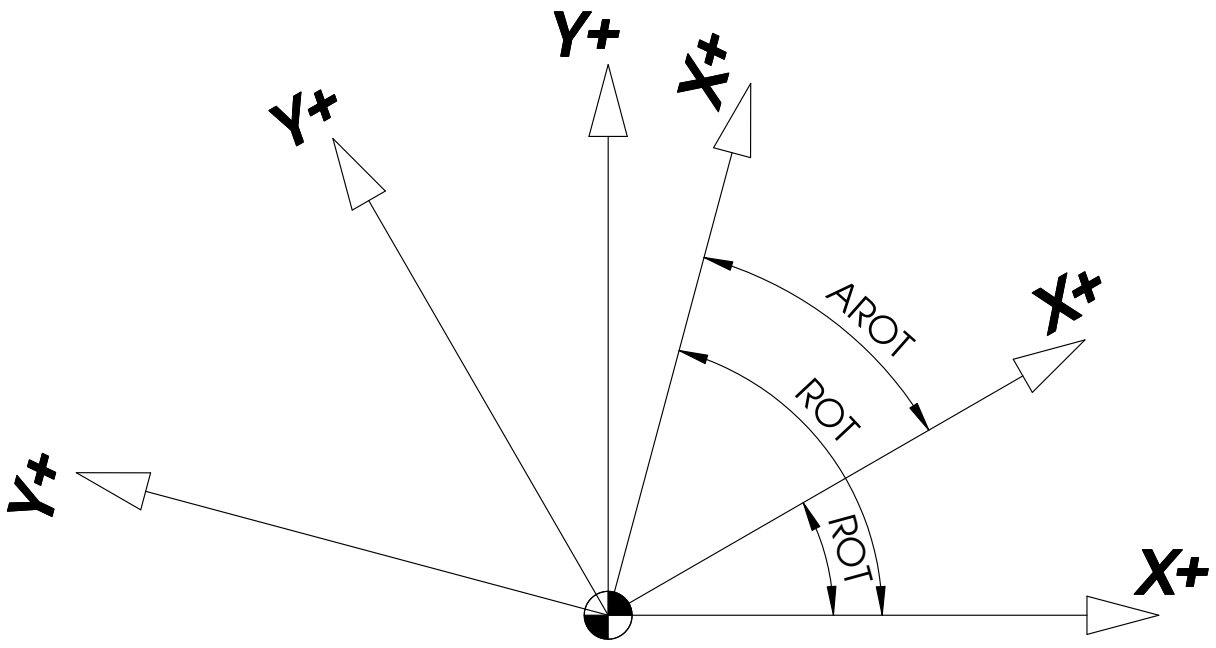
ROT X Y Z - The coordinate system is rotated through the programmed angle around the specified axes. The center of rotation is the last specified settable zero offset (G54 to G599).

AROT X Y Z - Rotation through the angle values programmed in the axis direction parameters.

Deactivate rotation: For all axes: ROT (without axis parameter)

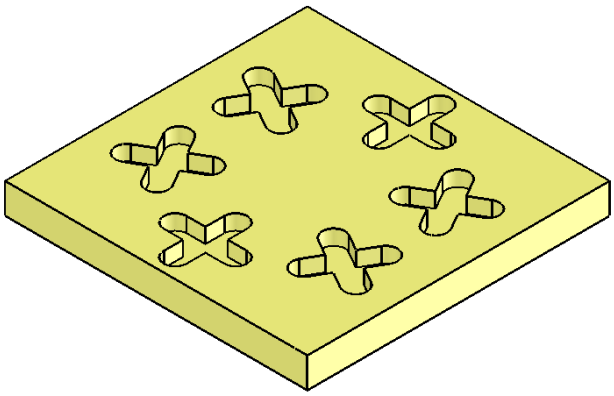
Syntax:

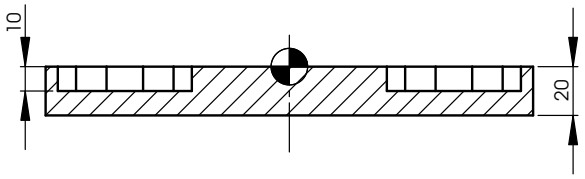
ROT RPL = ___

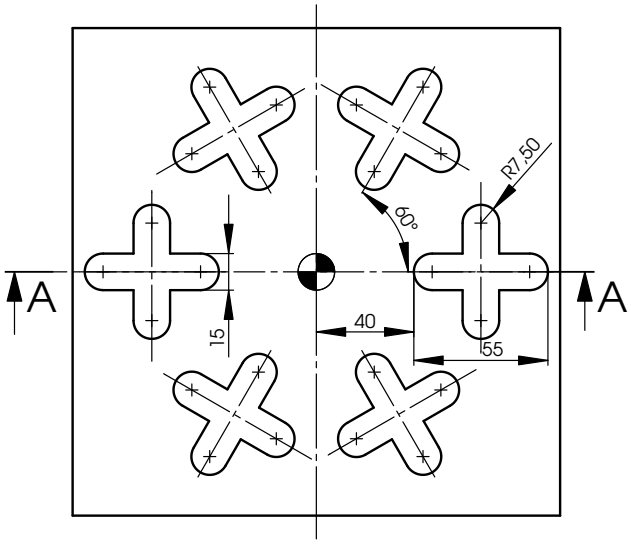


Example

```
G17 G71 G90 G94
G53 G0 Z-110 D0
T01;MILL D10
M6
G54 D01 G64 CFIN
S2000 M3
CRUZ P1
ROT RPL=60
CRUZ P1
ROT RPL=120
CRUZ P1
ROT RPL=180
CRUZ P1
ROT RPL=240
CRUZ P1
ROT RPL=300
CRUZ P1
ROT
G53 G0 Z-110 D0 M5
M30
```







SUB PROGRAM

CRUZ.SPF

```
G0 X67.5 Y0
Z5
G1 Z0 F500
STA: G1 Z=IC(-2.5) F200
G41 X75 Y-7.5 F600
X87.5
G3 Y7.5 CR=7.5
G1 X75
Y20
G3 X60 CR=7.5
G1 Y7.5
X47.5
G3 Y-7.5 CR=7.5
G1 X60
Y-20
G3 X75 CR=7.5
G1 Y-7.5
END: G40 X67.5 Y0
REPEAT STA END P3
G0 Z5
M17
```

20.3 - FUNCTION: SCALE, ASCALE

Programmable scale factor

SCALE/ASCALE enables you to program scaling factors in the direction of the axis specified for all path, synchronous and positioning axes. This enables the size of a shape to be changed. You can thus program similar geometrical shapes in different sizes.

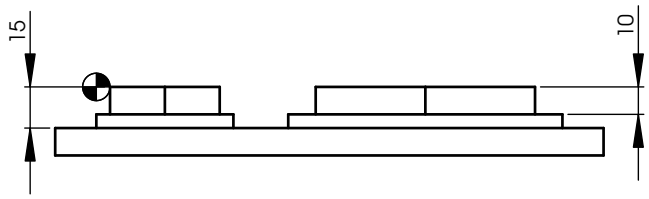
SCALE X Y Z - You can specify an individual scale factor for each axes, by which the shape is to be reduced or enlarged. The scale refers to the workpiece coordinate system set with G54 to G57. ASCALE X Y Z - You can program scale changes which are to be added to existing frames by using the ASCALE command. In this case, the last valid scale factor is multiplied by the new one.

Desactivate scaling factor: For all axes: SCALE (without axis parameter)

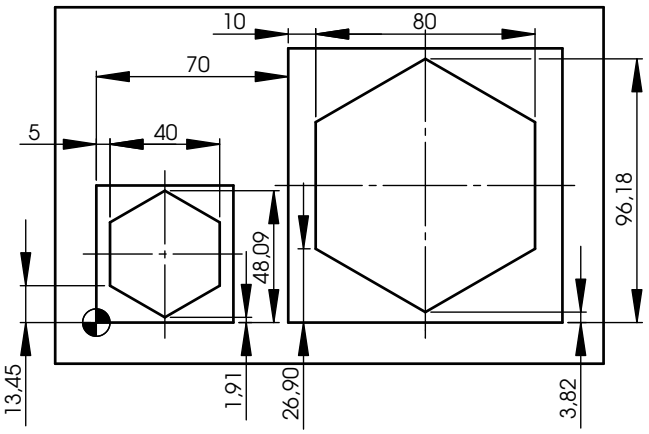
Syntax:

SCALE X___ Y___

Example:



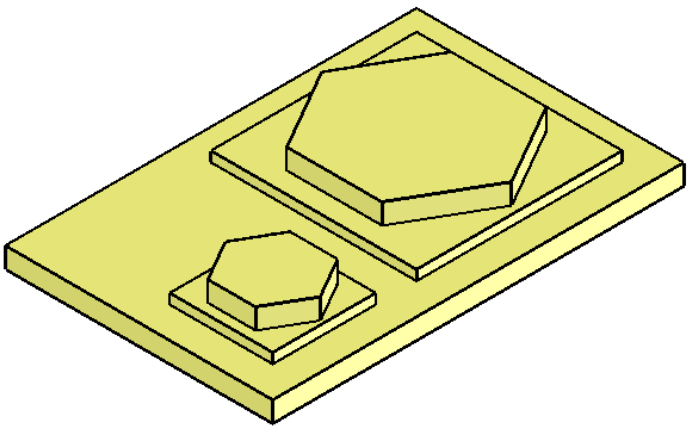

```
G17 G71 G90 G94
G53 G0 Z-110 D0
T01;MILL D20
M6
G54 D01 G64 CFIN S2000 M3
HEXAGON P1
TRANS X70
ASCALE X2 Y2
HEXAGON P1
TRANS (OR SCALE)
G53 G0 Z-110 D0 M5
M30
```



SUB PROGRAM

HEXAGON.SPF

```
G0 X25 Y-15
Z5
G1 Z0 F500
STA: G1 Z=IC(-2.5) F200
G41 X25 Y1.91 F600
X5 Y13.45
Y36.55
X25 Y48.09
X45 Y36.55
Y13.45
X25 Y1.91
END: G40 Y-15
REPEAT STA END P3
G0 Z5
M17
```



20.4 - FUNCTION: MIRROR, AMIRROR

Programmable mirror image,

MIRROR/AMIRROR can be used to mirror workpiece shapes on coordinate axes. All traversing movements which are programmed after the mirror call, e.g. in the subprogram, are executed in mirror image.

MIRROR X Y Z - The mirror is programmed by means of an axial change of direction in the selected working plan.

AMIRROR X Y Z A - mirror image which is to be added to an existing transformation is programmed with AMIRROR.

The mirror command causes the control to automatically change the path compensation commands (G41/G42 or G42/G41) according to the new machining direction.

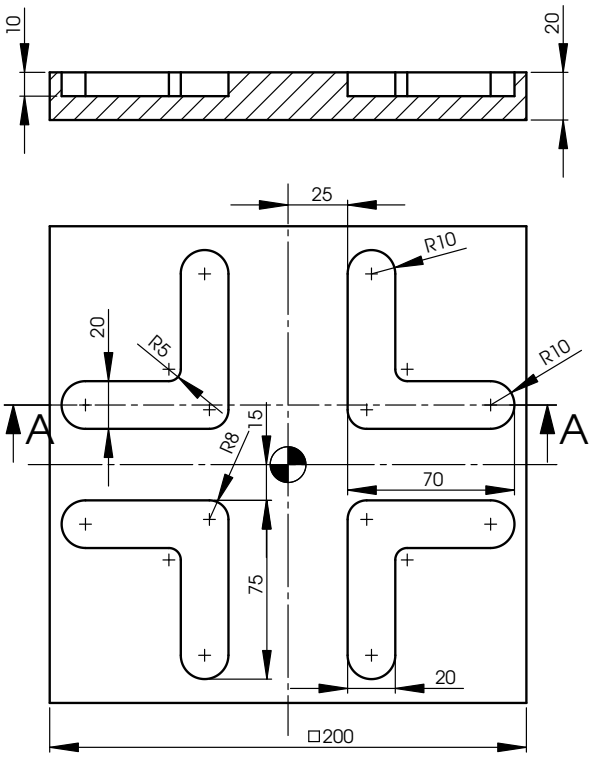
Deactivate mirror image: For all axes: MIRROR (without axis parameter)

Syntax:

MIRROR X__ Y___ Z___

Example:

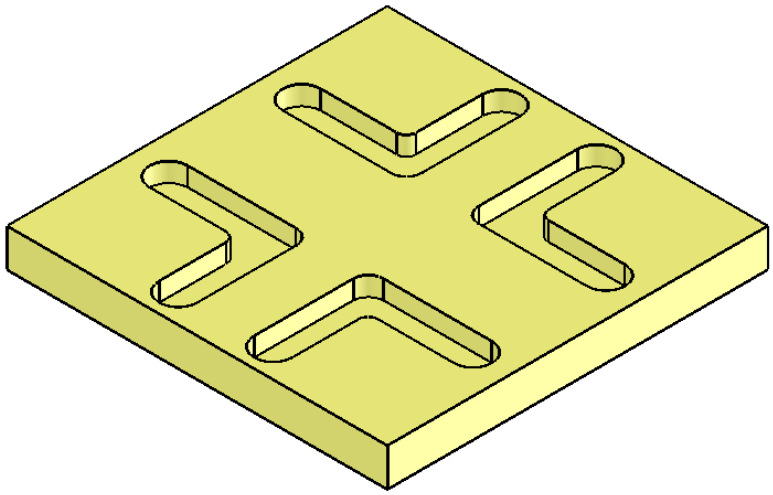
```
G17 G71 G90 G94
G53 G0 Z-110 D0
T01;MILL D15
M6
G54 D01G64 CFIN
S2000 M3
CONTOUR P1
MIRROR X0
CONTOUR P1
AMIRROR Y0
;or MIRROR X0 Y0
CONTOUR P1
MIRROR Y0
;or AMIRROR X0
CONTOUR P1
MIRROR
G53 G0 Z-110 D0 M5
M30
```



SUB PROGRAM

CONTOUR.SPF

```
G0 X35 Y25
Z5
G1 Z0 F500
STA: G1 Z=IC(-2) F200
G41 X33 Y15 F600
X85
G3 Y35 CR=10
G1 X45 RND=5
Y80
G3 X25 CR=10
G1 Y23
G3 X33 Y15 CR=8
END: G40 G1 X35 Y25
REPEAT STA END P4
G0 Z5
M17
```

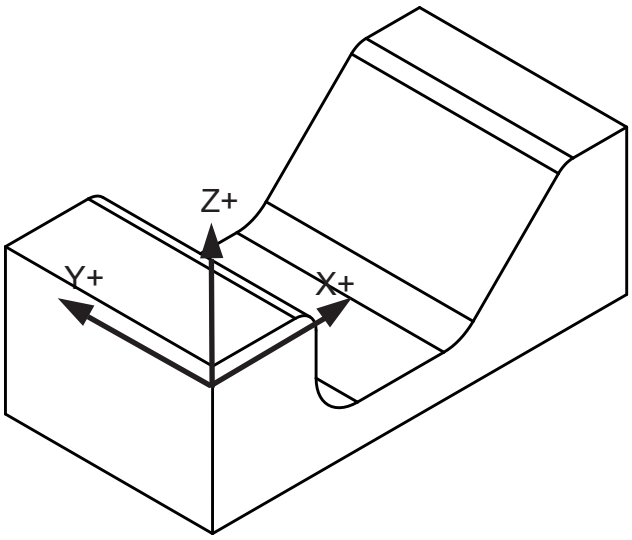


21 - WORKPIECE POINTS

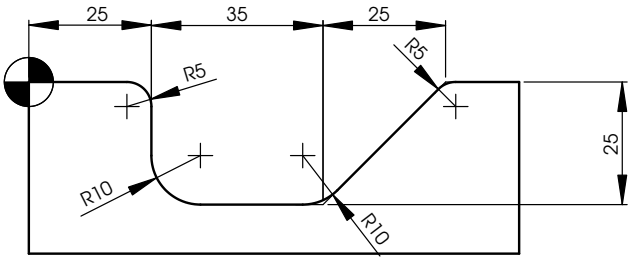
A plan is defined by means of two coordinate axes. The third coordinate axis is perpendicular to this plan and determines the infeed direction of the tool . When programming, it is necessary to specify the working plan in order that the control can calculate the tool offset values correctly. The plan is also relevant to certain types of circular programming and polar coordinates.

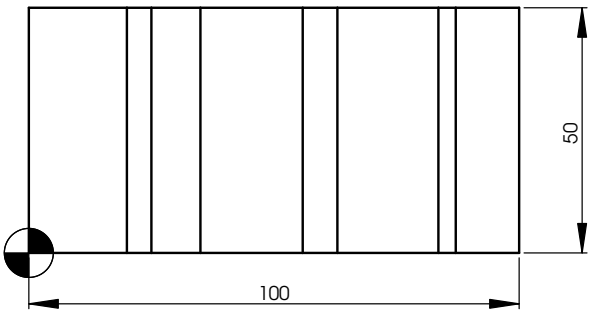
NC program with G17, G18 and G19:

Example G18:



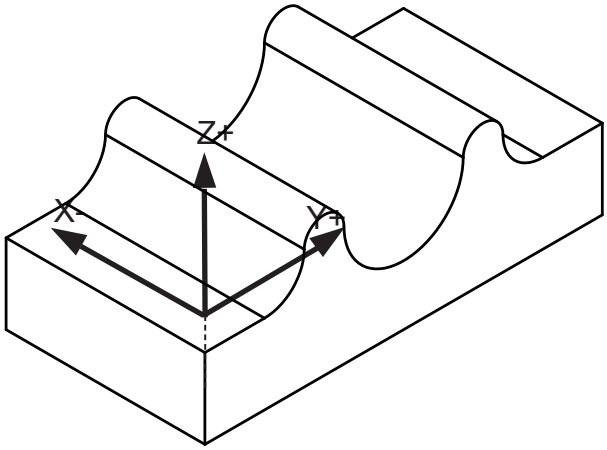

```
G18 G71 G90 G94
G53 G0 Z-110 D0
T1;BALL NOOSE. D8 MM
M6
G54 D1 S3600 M3
G64 CFIN
G0 X-10 Y0
Z10
AAA: G1 Y=IC(0.2) F360
G42 Z0
X20
G3 X25 Z-5 CR=5
;or G3 X25 Z-5 I=AC(20) K=AC(-5)
G1 Z-15
G2 X35 Z-25 CR=10
;or G2 X35 Z-25 I=AC(35) K=AC(-15)
G1 X60 RND=10
X85 Z0 RND=5
X110
BBB: G40 G0 X-10 Z10
REPEAT AAA BBB P249
G53 G0 Z-110 D0
M30
```

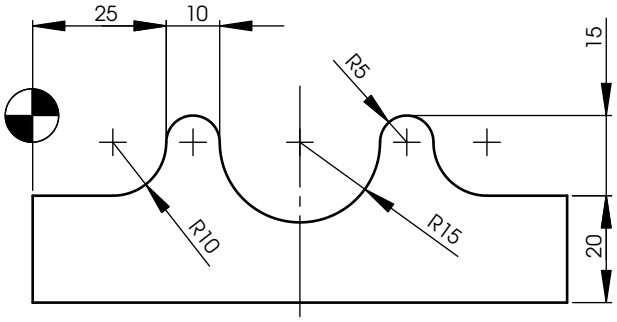


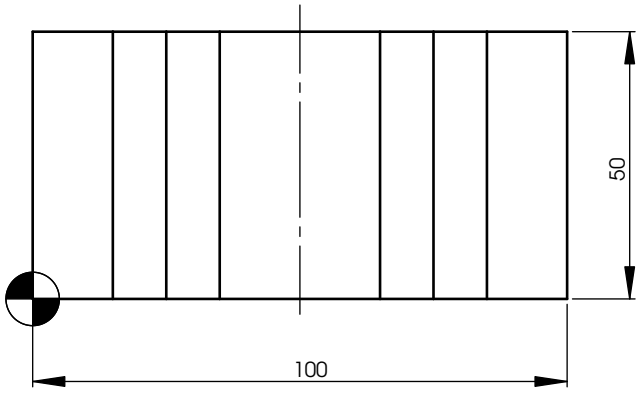


Example G19:

```
G19 G71 G90 G94
G53 G0 Z-110 D0
T1; BALL NOOSE D8 MM
M6
G54 D1 S3600 M3
G64 CFIN
G0 X0 Y-10
Z10
AAA: G1 X=IC(0.2) F360
G41 Z-15
Y15
G3 Y25 Z-5 CR=10
;or G3 Y25 Z-5 J=AC(15) K=AC(-5)
G2 Y35 CR=5
:or G2 Y35 J=AC(30) K=AC(-5)
G3 Y65 CR=15
;or G3 Y65 J=AC(50) K=AC(-5)
G2 Y75 CR=5
;or G2 Y75 J=AC(70) K=AC(-5)
G3 Y85 Z-15 CR=10
;or G3 Y85 Z-15 J=AC(85) K=AC(-5)
G1 Y110
G40 G0 Z10
BBB: Y-10
REPEAT AAA BBB P249
G53 G0 Z-110 D0
M30
```







NOTE:

The tool length components can be calculated according to the rotated working planes with the functions for "Tool length compensation for orientable tools". For more information on this calculation type, please refer to Chapter "Tool Offsets". The offset plan is selected with CUT2D, CUT2DF.

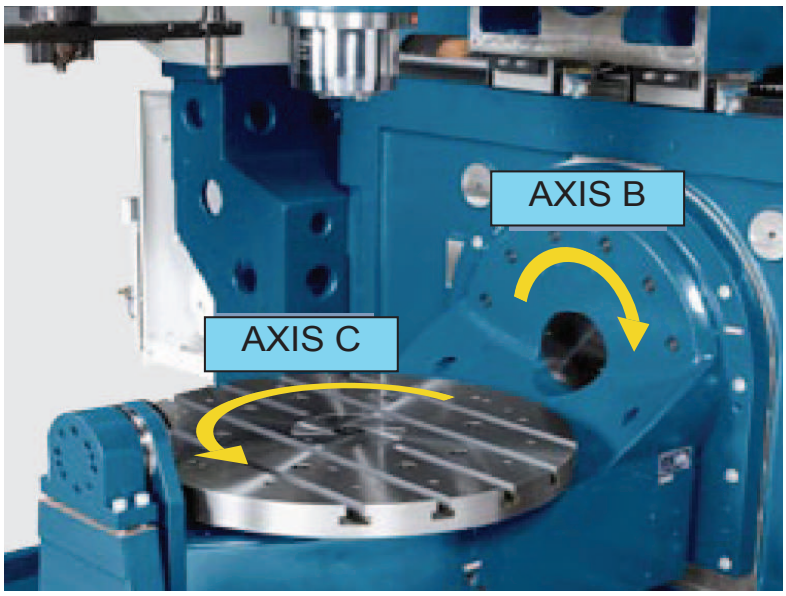
To happen the tool radius compensation, the used tool in this kind of machining should be a ball nose tool and the preset should be done for the radius center.

This kind of programming usually is used in finishing operations.

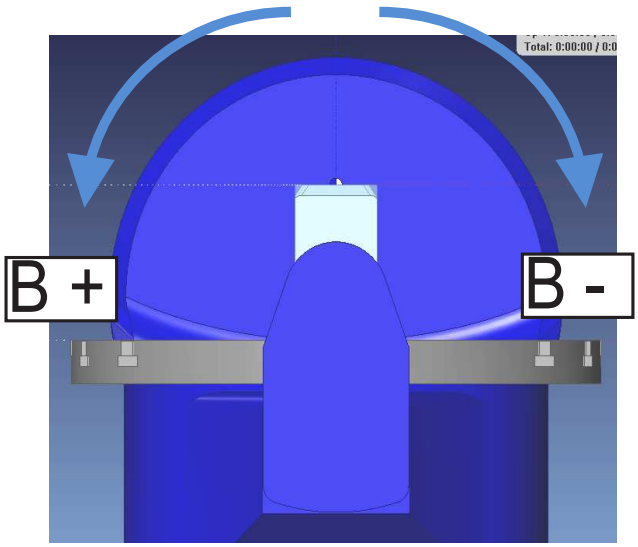
22 - ROTARY TABLE (AXIS B / AXIS C)

Machines from Line DCM 620 5X differ from an ordinary machining center, because they have two supplementary displacement axis, which allow more complex work. These axes are arranged in a setting known as “saddle”, and their indexing can make plans which would be impossible to be reached in ordinary machining centers.

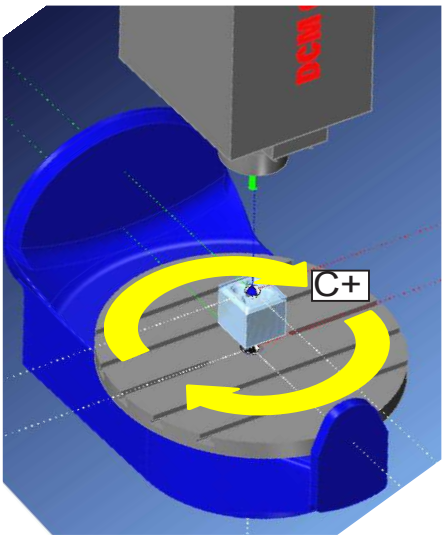
Additional axes (AXIS B and AXIS C) are set in degrees, in millesimal mode (3.3), both in positive direction and negative direction, obeying a limit range of -50º to +110º for axis B and a settable limit of 360º for axis C. With specific displacements, it is possible to generate several work situations, which shall be split in two cases: *WORK WITH 5 FACES and WORK WITH 5 AXES.*



DEFINITIONS OF AXES



FRONTAL SECTION (AXIS B)



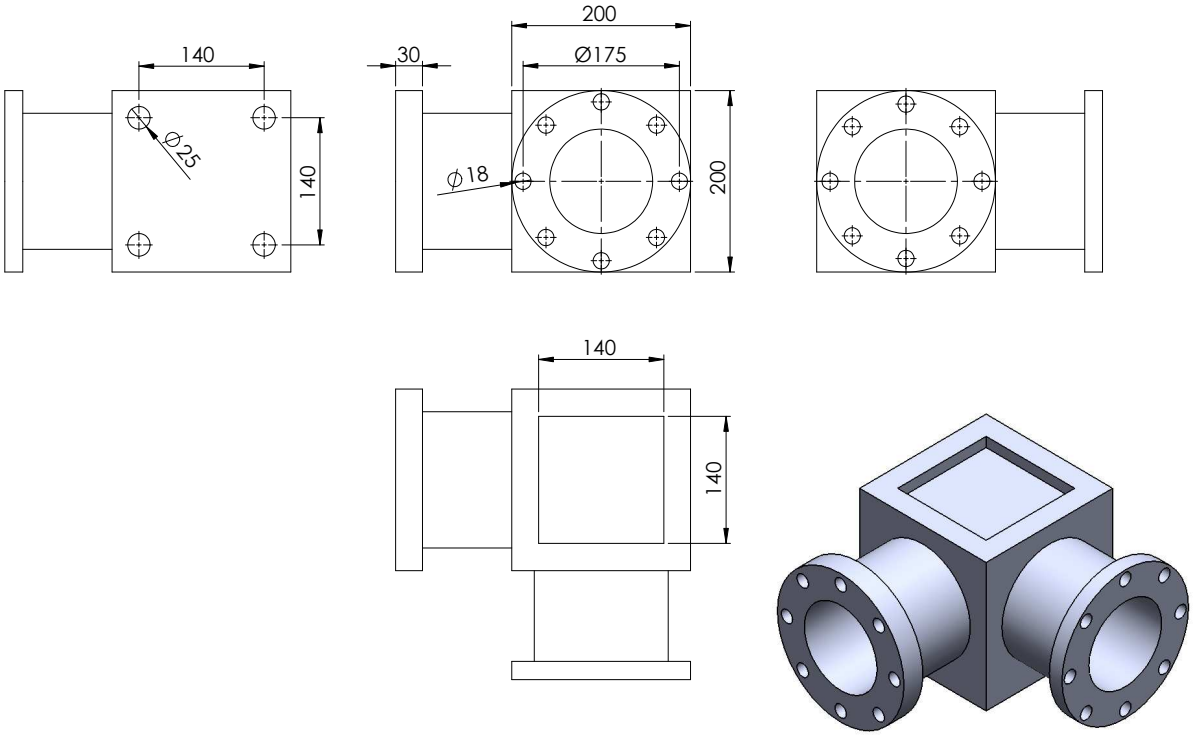
ISOMETRIC SECTION (AXIS C)

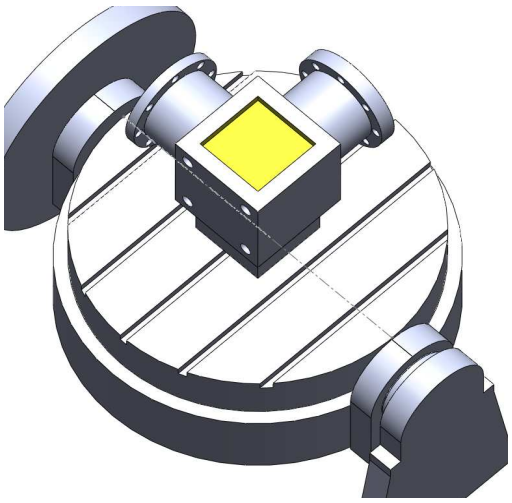
22.1 - WORK WITH 5 FACES

In these conditions, we consider a type of work where the part has several faces needing machining. For this, axes B and C shall be indexed in determined positions, where settings necessary for the respective work shall be applied. In this case, axes B and C trend to be static in the selected degree, and the machining of operations shall occur by displacing axes X, Y and Z, on which we can use all cycles already seen in this manual.

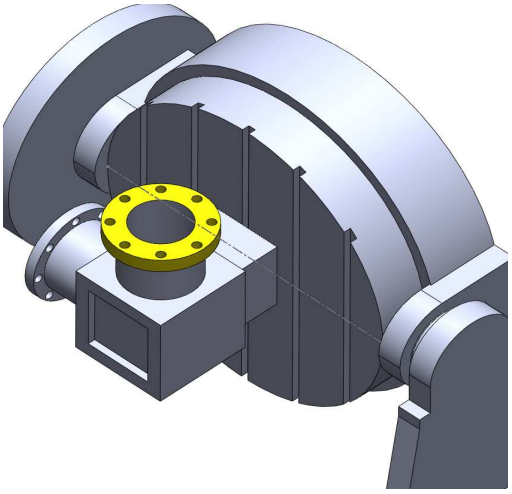
22.1.1 Examples of work using 5 FACES:

Perform machining of 5 faces of the example below:

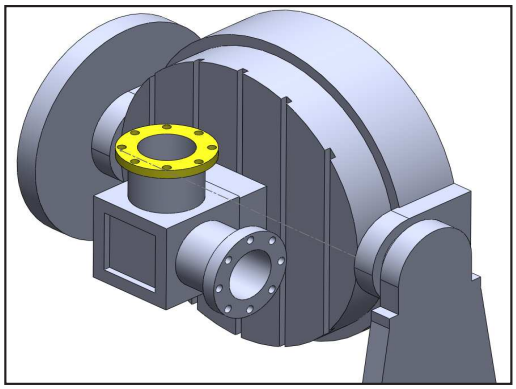




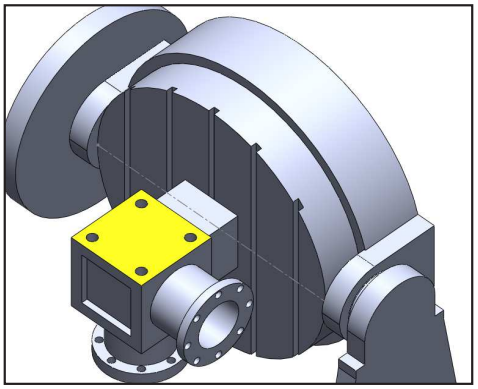
PHASE 1 – CAVITY (B0/C0) – G54



PHASE 2 – DRILLING (G90/C0) – G55



PHASE 3 – DRILLING (B90/C90) – G56



PHASE 3 – DRILLING (B90/C180) –G57

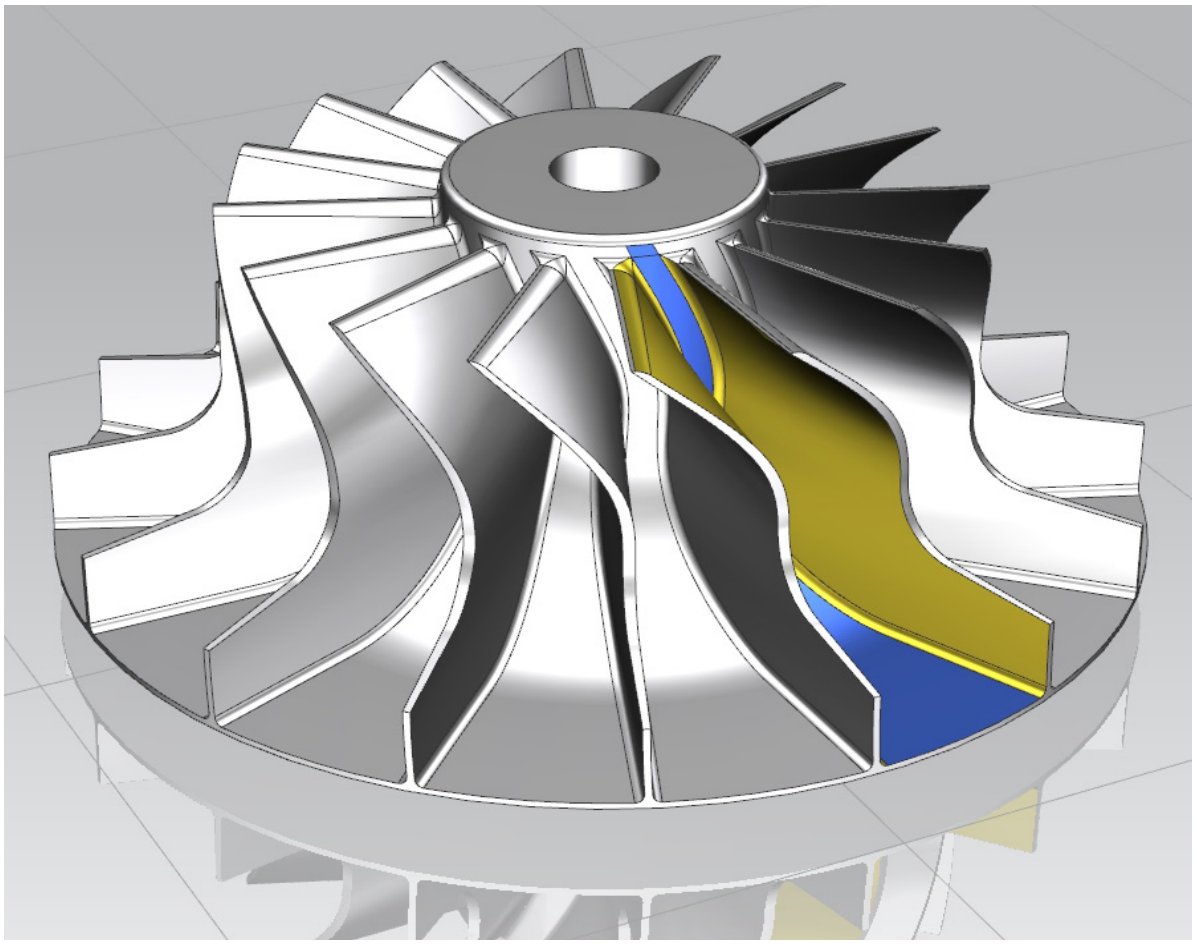
OBSERVATION: IN THIS CASE, A ZERO POINT WAS ATTRIBUTED FOR EACH SIDE OF THE PART.

Setting example:

MAIN.MPF N10 G17 G71 G90 G94 N20 G53 G0 Z0 D0 N30 T1; MILL D20 N40 M6 N50 **G54** D1 S3500 M3 N60 G0 X0 Y0 B0 C0 N70 G0 Z250 N80 PHASE_1 N90 G53 G0 Z0 D0 N100 T2; DRILL D18 N110 M6 N120 G55 D1 S1800 M3 N130 G0 X0 Y0 B90 C0 N140 G0 Z300 N150 **PHASE2_3** N160 G0 X-300 Z300 N170 G0 B90 C90 N180 **G56** G0 X0 Y0 N190 PHASE 2_3 N200 G0 X-300 X300 N210 G0 B90 C180 N200 **G57** G0 X0 Y0 N230 Z150 N240 PHASE4 N250 G53 G0 X0 Z0 D0 N260 G0 B0 C0 N270 M30	PHASE_1.SPF N10 POCKET3(110, 100,2,85,140,140,0,0,0,0,3,0,0,3000,250,0,11,10,15,10,2,0,2.5,0,0.5,-2,11100,11,0) N20 M17 PHASE2_3.SPF N10 F1000 N20 MCALL CYCLE83(300,230,2, 180,,220,,10,0,0,100,0,0,0,2,0,0.5,0,1,12121112) N30 POS1: HOLES2(0,0,87.5,0,40,8,1010,0,,,1) N40 MCALL N50 M17 PHASE4.SPF N10 F1000 N20 MCALL CYCLE83(150, 100,2,80,, 95,,10,0,0,100,0,0,0,2,0,0.5,0,1,12121112) X-70 T-70 X-70 Y70 X70 Y70 X70 Y-70 N30 MCALL N40 M17

22.2 WORK WITH 5 AXES

This work condition is applied for parts which have high level of complexity, where machining of some details shall be done by the simultaneous displacement of 3, 4 or 5 axes. This type of work needs the use of very complex programs, thus, the use of dedicated software is mandatory (CAD-CAM system).

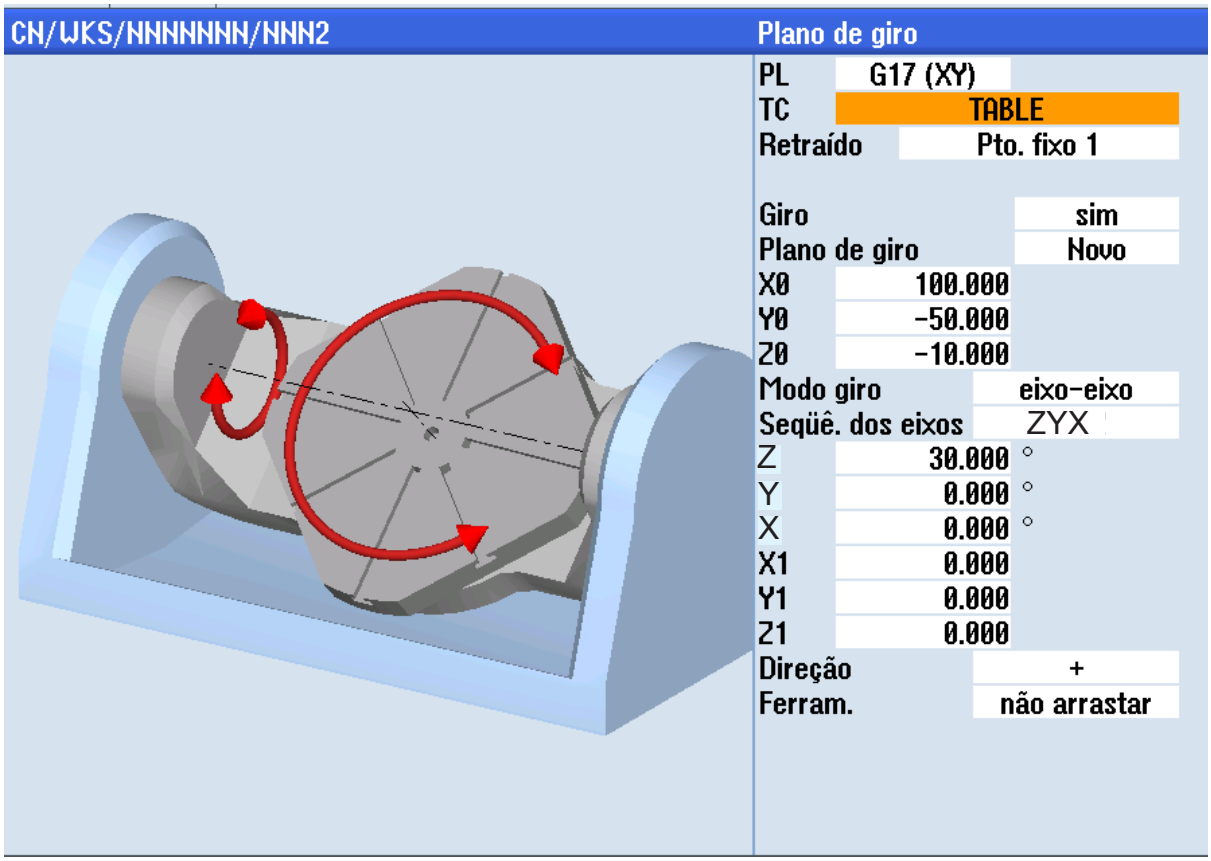


22.2.1 Cycle 800 (Revolution Plan)

This type of resource is more applied by CAM software, due to the complexity on manual setting, it allows a single Zero-Part and to work in the other faces making rotations and displacements of axes by field based on the initial zero.

In order to access the setting page of CYCLE 800, it is necessary to follow the following steps:

- Press softkey **[Various]**
- Press softkey **[Swivel Plane]**
- Fill the fields (see example figure below)
- Press softkey **[Accept]**



Syntax:

CYCLE800(__,TABLE,__,__,__,__,__,__,__,__,__,__,__,__,__,__)

Explanation:

Explanation:

PL	Rotation Plan
TC	TABLE = on / 0 = off
RETRACTED	MAX. TOOL DIRECTION = The tool retracts up to the highest point (G53) when turning the table. INCR. TOOL DIRECTION = The tool retracts the value contained in ZR field, when turning the table. NO = The tool not performs pitch when turning the table.
ROTATION	YES = The table turns when passing the cycle NO = The table not turns when passing the cycle, it is necessary to add a new approach.
ROTATION PLAN	NEW = An absolute plan is created ADDITIVE = An incremental plan is created.
X0	Displacement to a new zero point on axis X
Y0	Displacement to a new zero point on axis Y
Z0	Displacement to a new zero point on axis Z
TURNING MODE	In order to the machine use always **AXIS – AXIS**, due to its mechanical construction format.
SEQUENCE OF AXES	In order to the machine use always **ZYX**, due to its mechanical construction format.
Z	Axis rotation in relation to the new source (axis C)
Y	Axis rotation in relation to the new source (axis B)
X	There is no rotation on this axis (value 0)
Z1	Displacement of axis already rotated.
Y1	Displacement of axis already rotated.
X1	Displacement of axis already rotated.
DIRECTION	**+** = Positive direction of positioning revolution **-** = Negative direction of positioning revolution.
TOOLING	**DRAG =** Follows the tool tip when tilted. **NOT DRAG** = Not follows the tool tip when tilted

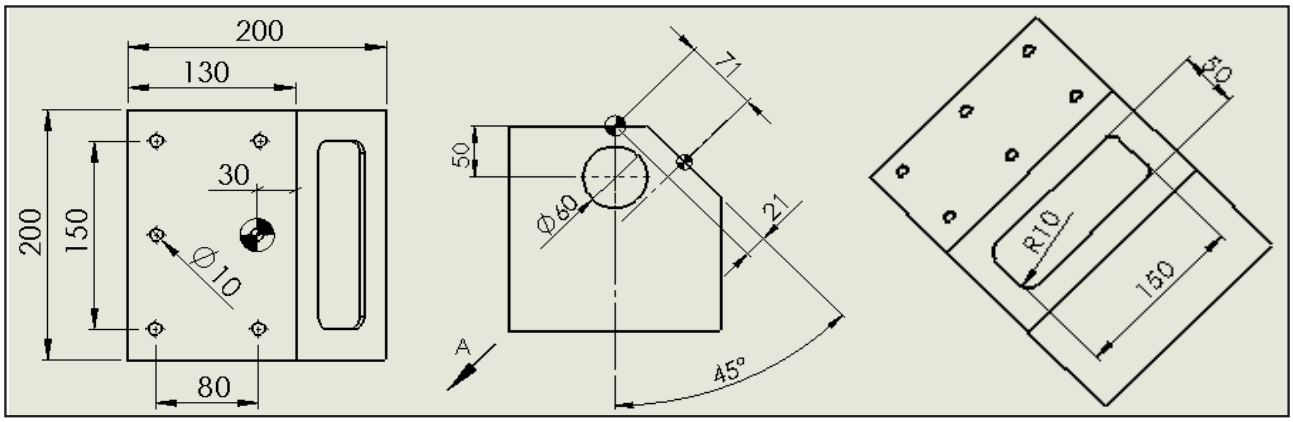
NOTES:

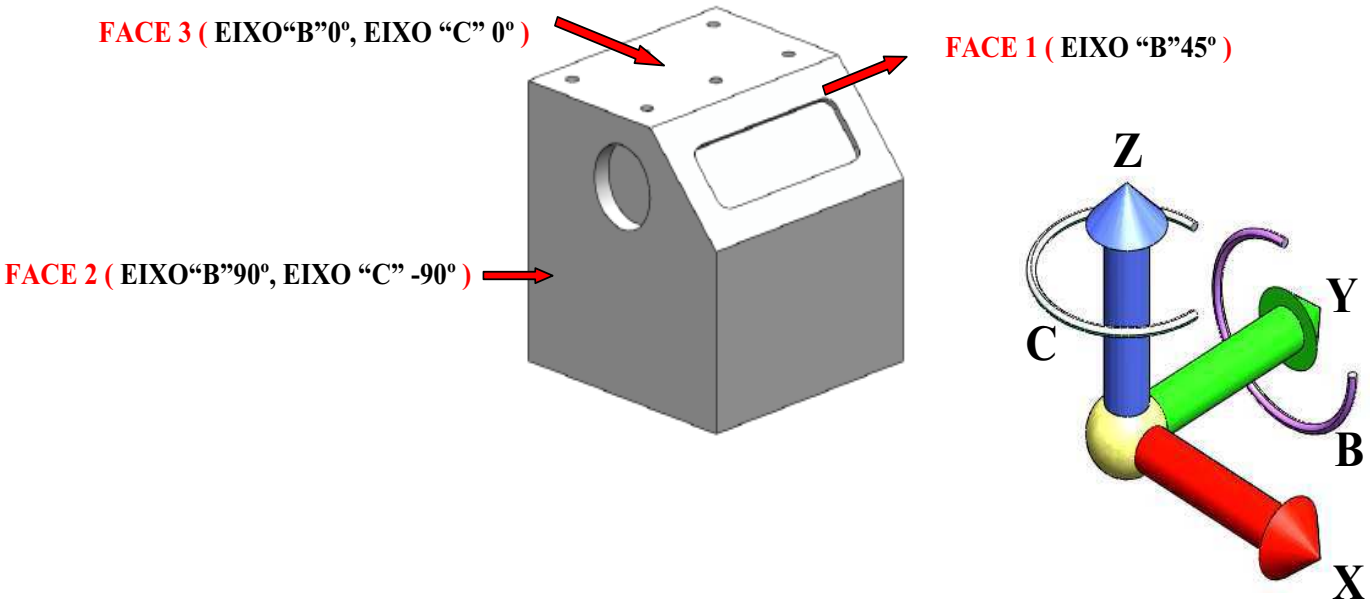
AFTER THIS RESOURCE IS APPLIED, IT IS NECESSARY TO DEACTIVATE THE FUNCTION, BECAUSE, ON THE CONTRARY, THE SYSTEM OF COORDINATES WILL BECOME ROTATED, ACCORDING THE LAST USE.

*IN ORDER TO DEACTIVATE,, JUST LEAVE **TC** FIELD WITH VALUE ZERO AND INSERT IN THE PROGRAM.*

22.2.2 Setting example

Execute the machining program of the model below, making the operations in the several faces with the aid of the function “CYCLE 800”.





MACHINING PROGRAM:

FACE “1”

```
N10 g17 g71 g90 g94
N20 WORKPIECE (,,,”RECTANGLE”,64,0—200,-80,200,200)
N30 G0 G53 Z0 D0
N40 T5;MILL TOP DIAM 20
N50 M6
N60 S2600 M3 D1
N65 G0 B0 C0
N70 CYCLE800(4,”TABLE”,200000,27,0,0,0,0,45,0,71,0,21,1,,1)
N80 CYCLE61(60,50,55,0,-71,-100,71,100,5,80,0,1200,31,0,1,11010)
N90POCKET3(60,0,55,-10,50,150,0,0,0,0,2.5,0,0,1200,0.1,0,21,75,8,3,15,5,2,0,1,2,11100,11,110)
```

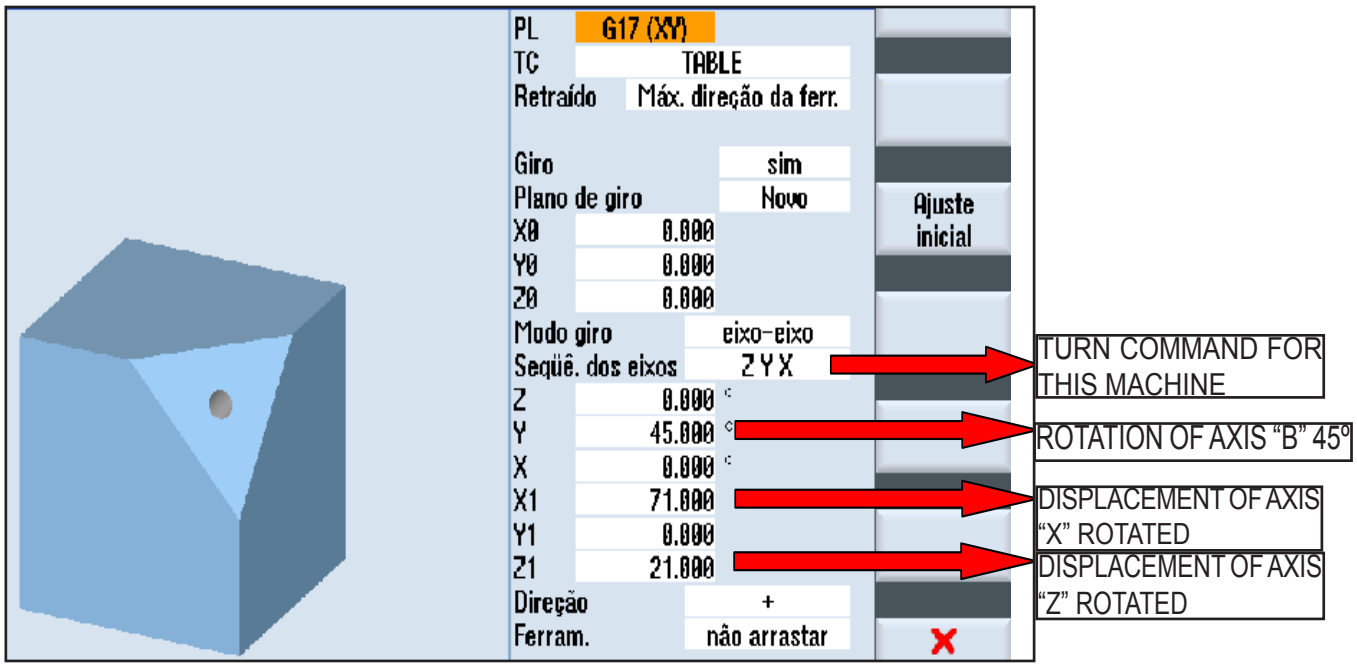


Table of cycle 800 set for the operation of face 1

FACE “2”

N100 CYCLE800(4,”TABLE”,200000,27,0,0,0,-90,90,0,0,0,100,1,,1)
N110 G0 X0 Y0
N120 POCKET (30,0,10,-10,
60,50,0,2,0,1200,0.1,0,1011,50,9,15,0,2,0,1,2,10100,111,110)

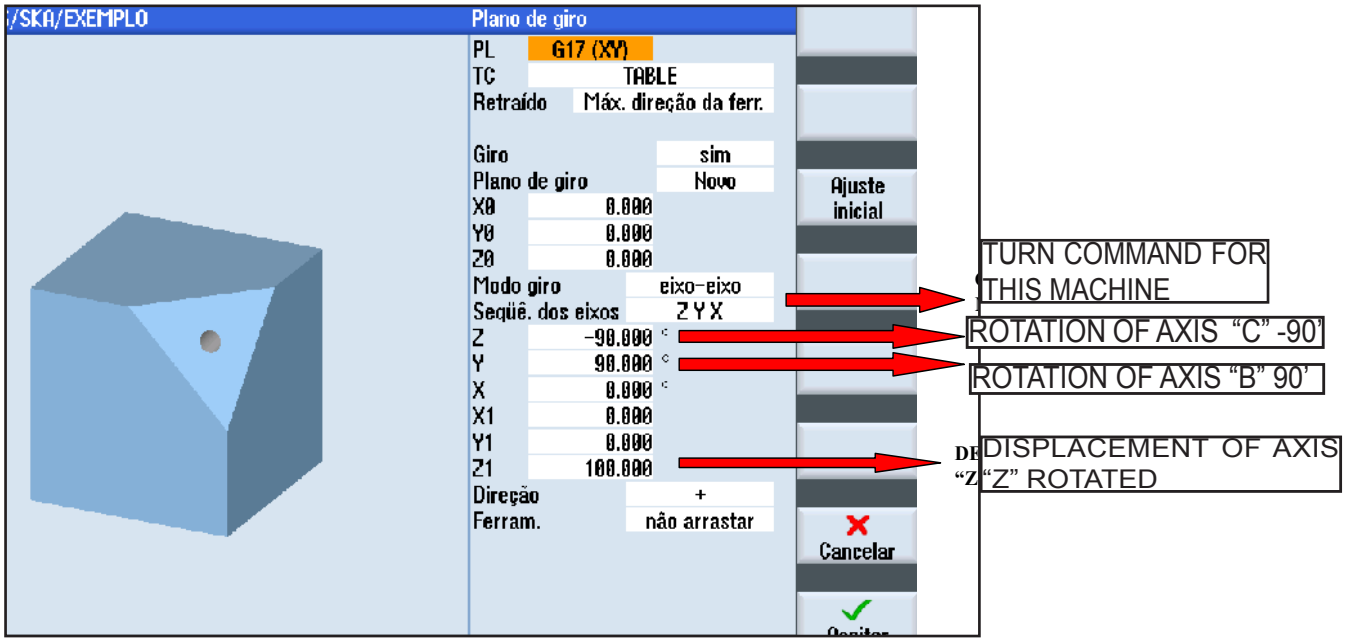


table of cycle 800 set for operation in face 2

FACE “3”

N150 CYCLE800(4,”TABLE”,200000,57,0,0,0,0,0,0,0,0,0,1,,1)
N160 CYCLE800()
N170 G0 G53 Z0 D0
N180 T7; DRILL DIAMETER 10
N190 M6
N200 S2600 M3 D1
N210 G0 X0 Y0
N220 F500
N230 MCALL CYCLE82(100,0,2,-25,,0,0,1,12)
N240 LABEL: CYCLE80(-80,-75,0,80,75,2,3,0,0,0,0,,,1)
N250 MCALL
N260 G0 G53 Z0 D0
N265 G0 B0 C0
N270 M30

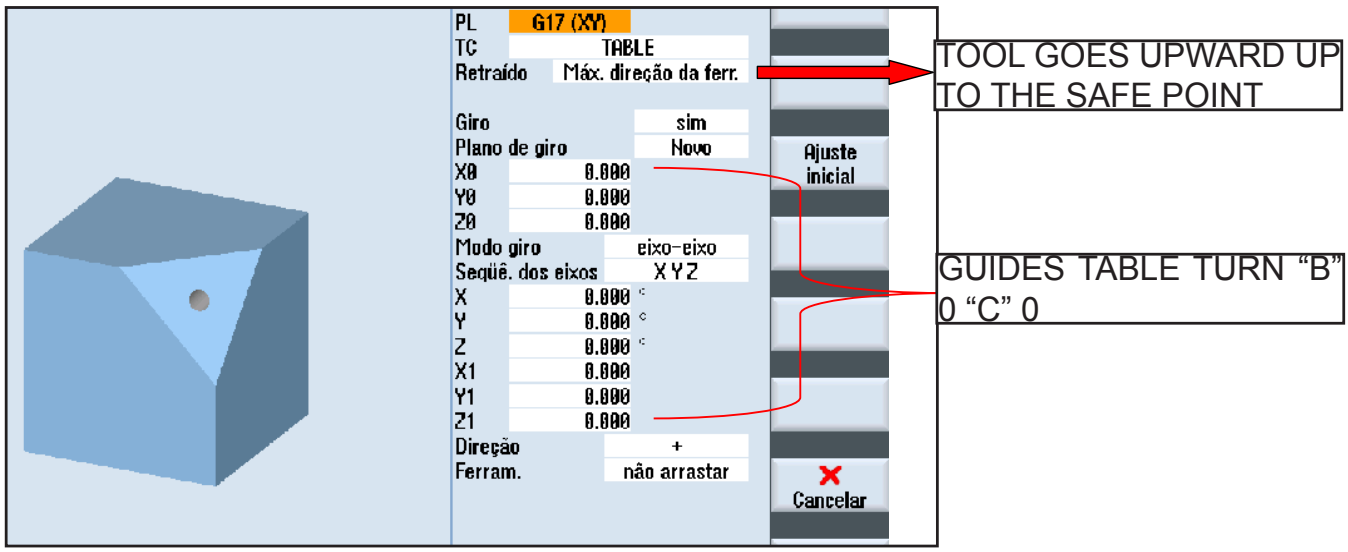
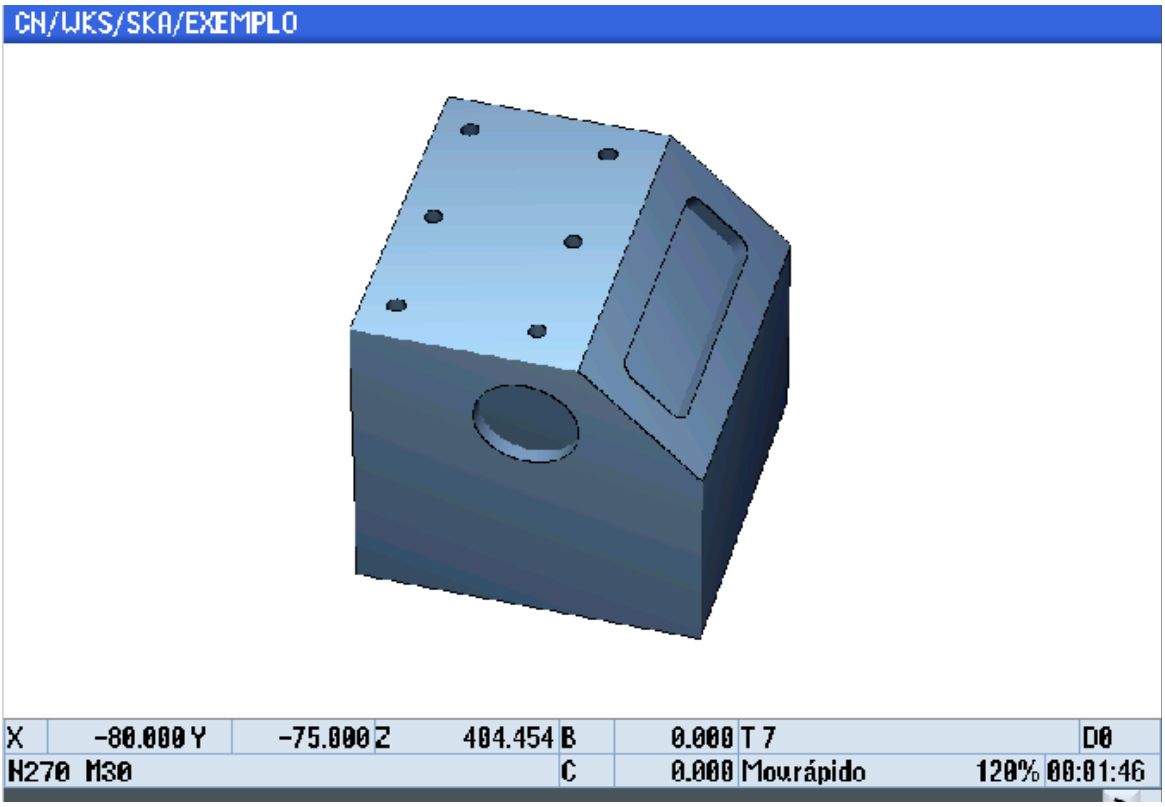


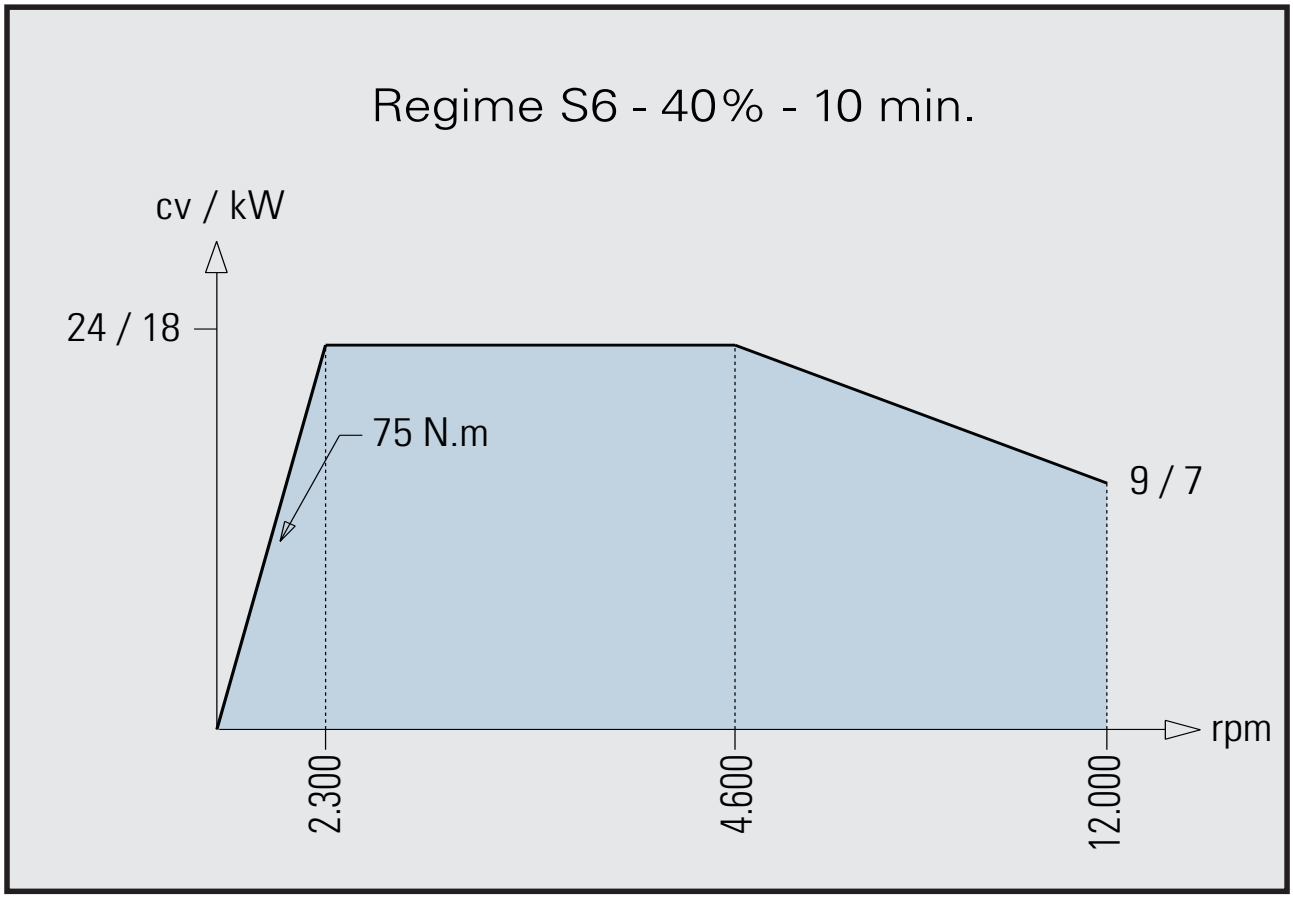
Table of cycle 800 set for operation on face 3

22.2.3 Graphic simulation



Example of graphic simulation screen

23 - POWER GRAPHIC



24- PROGRAMMING MOLDS AND DIES:

The Romi machining centers with SIEMENS 828 D has a optional to improve the machine performance when it's necessary to machining parts with complex profile where are done short and accurate moves keeping a good speed and acceleration.

This optional called "MOLDS AND DIES" is programmed with the "CYCLE 832" function and it has many specials software functions in the CNC that, associated to several machine parameters and hardware, allow:

- Better control of axes precision;
- High feedrates;
- Soft moves.

In this chapter will be explained the main function responsible for this better machines performance:

- Acceleration / Deceleration "Bell-Shape";
- Look-Ahead;
- SIEMENS tecnology for Mold and Die ("CYCLE832").

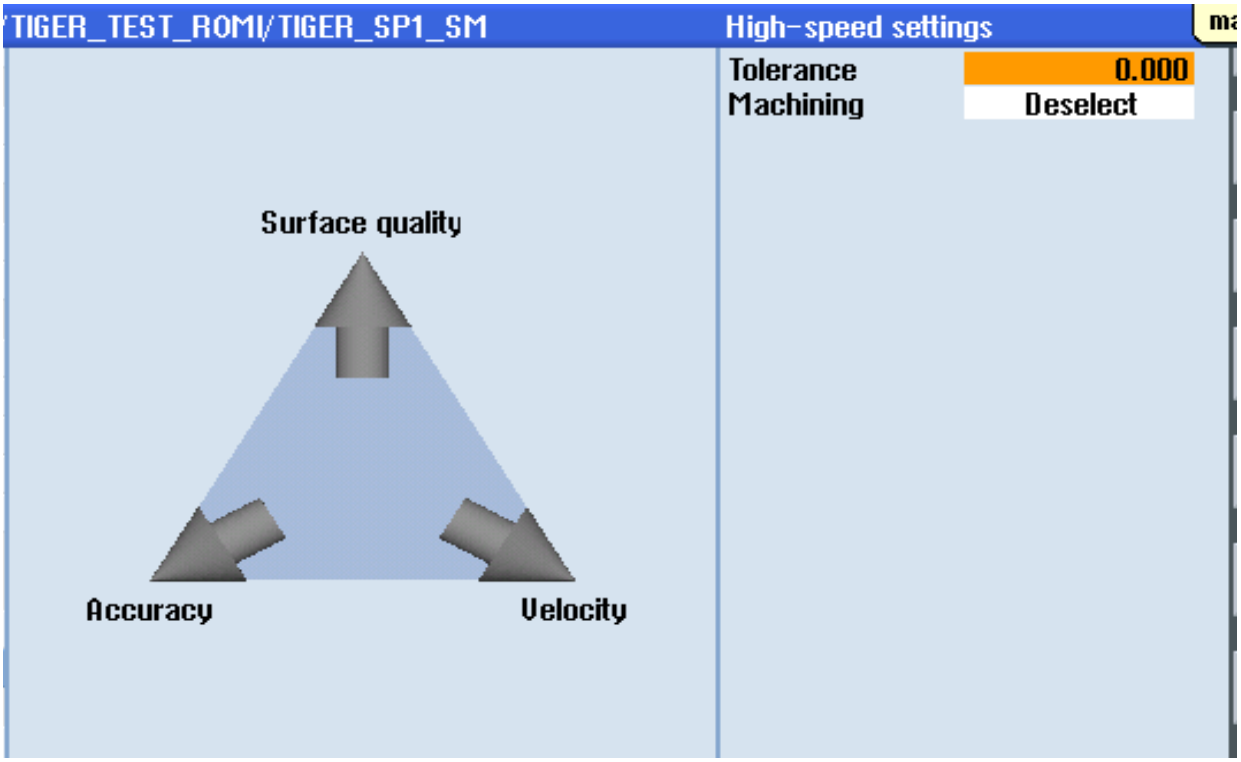
24.1 - INTERACTIVE SCREEN TO CONTROL THE MOLD AND DIE FUNCTIONS.

This screen allows the user to select, in a interactive way, which the functions he wants to apply during the machining of determined molds or dies.

When specifying which kind of machining to be performed (roughing, pre-finishing or finishing), the control automatically recognize the recommended functions for that kind of work, but, case be the user interest, they can be changed according with desired.

To access the **"CYCLE832"** screen, the user should:

- Press the **[Various]** horizontal softkey ;
- Press the **[HighSpeed Settings]** vertical softkey ;
- Fill de blanks (see the following picture)
- Press the **[Accept]** softkey ;



The following data should be filled:

Tolerance	This value should be 20 % bigger than CAM tolerance used to generate the program.
Type	Through of the **[SELECT]** key, choose among the options: -▼ (Roughing) -▼▼ (Pre - Finishing) -▼▼▼(Finishing)

After press the **[Accept]** softkey, the cycle is put in the program according the example below:

CYCLE832 (0.01, 2, 1)

To make the cycle programming out of machine, it means, without the interactive screen, the user should to program according syntax below:

CYCLE832(___, _ , _)

Explanation:

Tolerance	This value should be 20 % bigger than CAM tolerance used to generate the program.

Type	Through of the **[SELECT]** key, choose among the options: -1 (Roughing) -2 (Pre - Finishing) -3 (Finishing)
XXX	1 - To actives the cycle. 0 - To desactives the cycle.

24.2. Examples of CYCLE 832 programming:

ROUGHING: CYCLE 832 (0.1,1,1)

PRE-FINISHING: CYCLE 832 (0.05,2,1)

FINISHING: CYCLE 832 (0.02,3,1)

In a program, the CYCLE 832 can be inserted in the begin according example below:

```
N10 CYCLE832 (0.1,1,1)
N20 G17 G71 G90 G94
N30 G53 G0 Z0 D0
N40 T1
N50 M6
:
:
:
```

25 - "R" CALCULATION PARAMETER

This chapter has for objective to explain some special available resources in the CNC SIEMENS 828D that are considered more usual. The content is a synthesis of the original guide SIEMENS "FUNDAMENTAL BEGINNINGS - chapter 10" and "PROGRAMMING: ADVANCED - chapter 1".

25.1 - "R" CALCULATION PARAMETER

25.1.1 - Explanation:

"R" calculation parameter are fixed registers from R0 to R99 available to attribute values and are used like variable.

25.1.2 - Application:

Program development of family parts where the geometry are the same, but, with variable dimension values, two-dimensional profiles development and / or tridimensional generated step by step where the coordinates are calculated through contained logics in the program with conditional deviation, etc.

25.1.3 - Values attribution:

To the "R" parameters can be attributed directs and indirect values which the result should be in de following range:

+ or - (0.0000001 - 9999.9999) (eight decimal houses + signal + decimal point)

For entire values, the decimal point can be omitted.

A positive signal can be omitted.

Example:
R0=3.5678
R1=-36.4
R3=-3.65
R4=-6765.1234

Through of exponential notation "EX" can be allocated a enlarged numbers range, then, the exponential value is written after the EX characters.

Example:
R0=-0.1EX-5 Meaning: R0=-0.000 001
R1=1.874EX8 Meaning: R1=187 400 000

NOTE: *It's allowed to make a series of values attributions in a unique block, but, the block should be exclusive for attribution.*

Right example: R0=3.5678 R1=-36.4 R3=-3.65EX3 R4=3.23EX-2

Wrong example: R7=15 R15=3.1415 R42=R15*(COS(R7))

25.1.4 - Parameter application in a program:

The calculation parameter and / or arithmetic expressions with calculation parameter can substitute values in all program address except N, G and L, for this, is written after the address character "=" the parameter identification, followed or not of a arithmetic expression.

Examples:

N10 R5=24 R10=250
Efect: Atributed value 24 to R5 parameter and 250 to R10 parameter.

N20 G1 X=R5 F=R10
Efect: The machine will move until the coordinate X24 with feedrate=250 mm/min

25.1.5 - Operation / arithmetic functions:

When is applied the calculation / arithmetic function it's necessary to note the usual mathematics notation, it's means, the multiplication or division has the priority on addition or subtraction. To define the arbitrary priority and / or specific of the execution, we use the "parentheses". In all circumstances where degree values are used, the same should be declared in millesimal values.

Examples:

N60 R1=8 R20=SIN(30.345) R9=R7*R8 R12=R10/R11
N70 R13=R1*R20-R9
N80 R15=SQRT(R13+R9*R1)

25.2 - OPERATORS / ARITHMETIC FUNCTIONS

25.2.1- Main operators and arithmetic functions:

The "R" calculation parameters, like we saw in the before chapter, can be submitted to many kinds of calculation. The main operators are:

Operators	Function
+	Addition
-	Subtraction
*	Multiplication
/	Division
SIN(*)	Sine
COS(*)	Cosine
TAN(*)	Tangent
ASIN(*)	Arc sine
ACOS(*)	Arc cosine
ATAN2(*)	Arc tangent
SQRT(*)	Square root
ABS(*)	Absolute number
POT(*)	Potency of 2 (square elevation)
TRUNC(*)	Trunc to entire
ROUND(*)	Round to entire
LN(*)	Natural logarithmic
EXP(*)	Exponential function

* = insert value

Examples:

R1=R1+2 Result: Value that is in R1 + 2.
R3=SIN(30) Result: R3 will have the value sine of 30.
R4=R1+R20/R3 Result: R4 will have the value of equation
R5=(R1+R20)/R3 Result: R5 will have the value of equation

25.2.2 - Arithmetic function ATAN2(___,___)

This function calculated the result vector angle (hypotenuse) having as data others 2 vectors (cathetuses) of a straight line angle. The result will consider one of 4 quadrants (-180 < 0 < 180º). The angular reference is always based in a line started in the vertex of de 2nd value followed in a positive direction.

Examples:

R20=ATAN2(30.5,80.1) Result: 20.8455 degree
R21=ATAN2(30.5,-80.1) Result: 159.1545 degree
R22=ATAN2(-30.5,-80.1) Result: 200.8455 degree
R23=ATAN2(-30.5,80.1) Result: 339.1545 degree

25.3 - OPERATORS AND LOGIC COMPARATORS

25.3.1 - Comparison operators:

The comparison operators can be used to formulate a deviation condition. Complex expressions can also be compared. Look below:

Operators	Function	Description
= =	Equal to	Equal to
< >	Not equal to	Not equal to
>	Greater than	Greater than
<	Less than	Less than
>=	Greater than or equal to	Greater than or equal to
< =	Less than or equal to	Less than or equal to

25.3.2 - Logic operators:

Logic operators are used to check the true or false condition in a comparison between 2 values making a conditional deviation.

Syntax:

IF (comparison) GOTOB (LABEL DESTINE)
IF (comparison) GOTOF (LABEL DESTINE)

***NOTE**: The deviation can be to a block (LABEL) that is to back or ahead from de current conditional block. If it's to back then use GOTOB (GO TO BACKWARD), if it's ahead GOTOF (GO TO FORWARD).*

LABEL DESTINE is the identification of the block for which the execution should be deviated in case of a true result in the comparison.

In the case of false result in the comparison, the deviation isn't done, then, the execution will continue in the next block.

Example 1:

IF R10>=R11 GOTOB START

In the case of R10 be bigger or equal to R11, the execution will be deviated to the block LABEL called like START that is programmed to back from this block.

Example 2:

IF R20= = (SIN(R31)) GOTOF POSITION

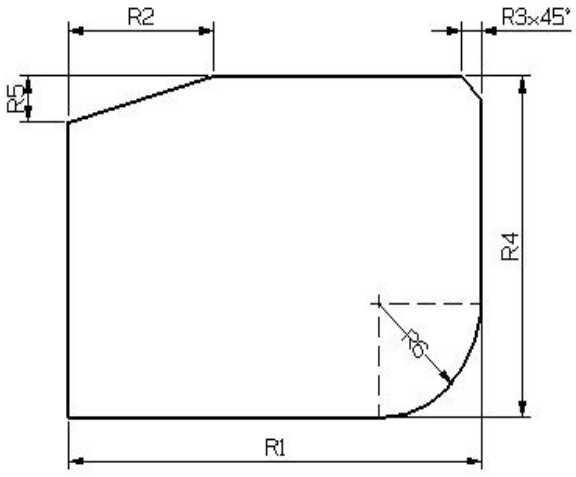
In the case of R20 to be equal to sine of R31 the execution will be deviated to the block (LABEL) called POSITION that is programmed after this block.

25.4 - PARAMETRIC PROGRAM EXAMPLES.

Example 1:

To elaborate a parametric program to a part family, according with the draw below:

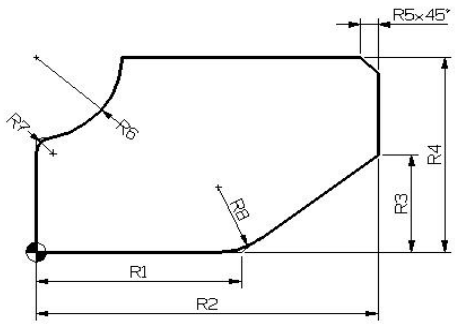
```
 %_N_EXE_2_MPF
;$PATH=/_N_WKS_DIR/_N_EXAMPLE_WPD
G17 G64 G71 G90 G94
G53 G0 Z-110 D0
T2
M6
G54 D1 S2500 M3
R1=200;                 TOTAL LENGTH OF THE PART
R2=70;                  LENGTH OF “CHAMFER” 1
R3=10;                  CHAMFER 2 (45 DEGREE)
R4=150;                 TOTAL WIDTH OF THE PART
R5=20;                  WIDTH OF “CHAMFER” 1
R6=50;                  RADIUS
R7=10;                  TOOL DIAMETER
R8=3;                   SECURITY DISTANCE
R9=350;                 FEEDRATE
R7=R7/2;                RADIUS TOOL CALCULATION
G0 X=-(R7+R8) Y=-(R7+R8)
Z5
Z-5
G42 G1 X0 Y0 F=R9
X=R1-R6
G3 X=R1 Y=R6 CR=R6
G1 Y=R4-R3
X=R1-R3 Y=R4
X=R2
X0 Y=R4-R5
Y0
G40 X=-(R7+R8) Y=-(R7+R8)
G53 G0 Z-110 D0 M5
M30
```



Example 3:

Elaborate a parametric program to a part family, according with the draw below:

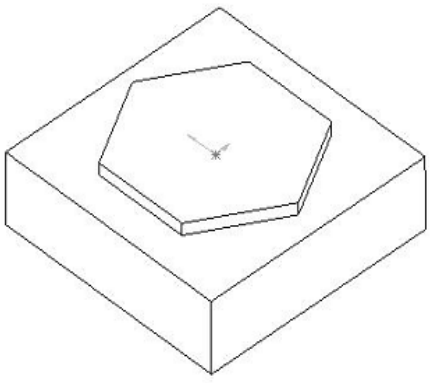
```
 %_N_EXE_2_MPF
;$PATH=/_N_WKS_DIR/_N_EXAMPLE_WPD
G17 G64 G71 G90 G94
G53 G0 Z-110 D0
T3
M6
G54 D1 S3500 M3
R1=60;                  LENGTH 1
R2=100;                 TOTAL LENGTH OF THE PART
R3=30;                  WIDTH 1
R4=60;                  TOTAL WIDTH OF THE PART
R5=5;                   CHAMFER WIDTH x 45 DEGREE
R6=25;                  RADIUS 1
R7=7;                   RADIUS 2
R8=20;                  RADIUS 3
R9=20;                  TOOL DIAMETER
R10=400;                FEEDRATE
R11=0;                  START Z
R12=-20;                END Z
R13=2;                  CUT DEPTH
R14=5;                  SECURITY DISTANCE IN X AND Y
R15=5;                  SECURITY DISTANCE IN Z
R9=R9/2;                TOOL RADIUS CALCULATION
R16=R11+R15;            START Z CALCULATION
R20=R11-R13;            FIRST CUT DEPTH COORDINATE
G0 X=-(R9+R14) Y=-(R9+R14)
Z=R16
AAA: G0 Z=R20
BBB: G42 G1 X0 Y0 F=R10
X=R1 RND=R8
X=R2 Y=R3
Y=R4 CHR=R5
X=R6
G2 X0 Y=R4-R6 CR=R6 RND=R7
G1 Y0
CCC: G40 X=-(R9+R14) Y=-(R9+R14)
R20=R20-R13
IF R20>R12 GOTOB AAA
G0 Z=R12
REPEAT BBB CCC
G0 Z=R16
G53 G0 Z-110 D0 M5
M30
```



Example 3:

Elaborate a parametric program to make a inscribed hexagon in a determined circle:

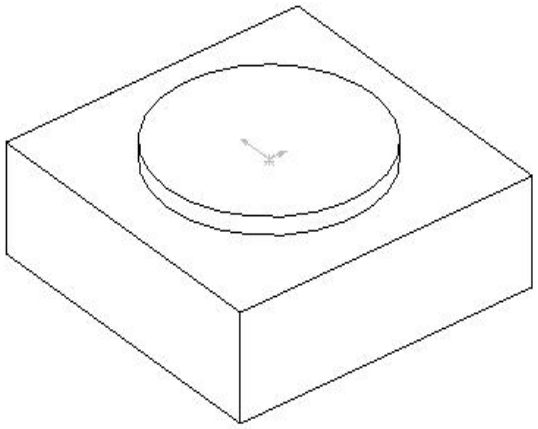
```
%_N_EXE_3_MPF
;$PATH=/_N_WKS_DIR/_N_EXAMPLE_WPD
G17 G64 G71 G90 G94
G53 G0 Z-110 D0
T12
M6
G54 D1 S2700 M3
R1=50;                          CIRCLE RADIUS
R2=10;                          TOOL DIAMETER
R3=0;                           START ANGLE
R4=3;                           SECURITY DISTANCE
R6=0;                           NUMBER OF SIDES ACCOUNTANT
R2=R2/2;                        TOOL RADIUS CALCULATION
R1=R1+R2;                       CIRCLE RADIUS REDEFINITION
R10=R3;                         R3 VARIABLE DUPLICATION
R11=200;                        FEEDRATE
G00 X=((R1+R4)*COS(R3)) Y=((R1+R4)*SIN(R3))
Z2
G01 Z-5 F150
START: G01 X=(R1*COS(R3)) Y=(R1*SIN(R3)) F=R11
R3=R3+60
R6=R6+1
IF R6<=6 GOTOB START
G1 X=((R1+R4)*COS(R10)) Y=((R1+R4)*SIN(R10))
G53 G00 Z-110 D0
M30
```



Example 4:

Elaborate a parametric program in order to execute arcs from 0,001 to 360 degrees of aberture using the G01 function.

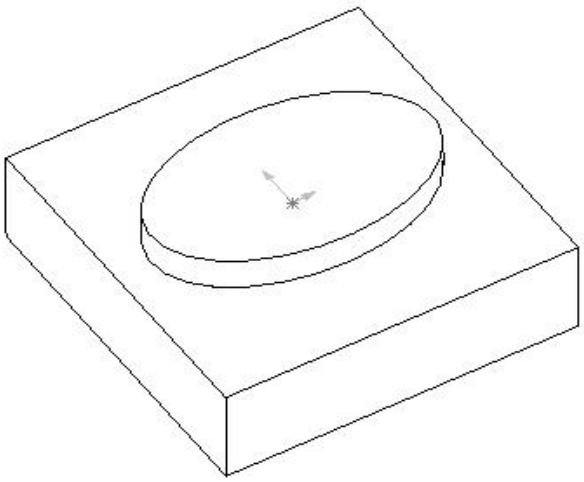
```
%_N_EXE_4_MPF
;$PATH=/_N_WKS_DIR/_N_EXAMPLE_WPD
G17 G64 G71 G90 G94
G53 G0 Z-110 D0
T1
M6
G54 D1 S2500 M3
R1=50;                          CIRCLE RADIUS
R20=10;                         TOOL DIAMETER
R3=0;                           START ANGLE
R4=360;                         END ANGLE
R5=1;                           INCREASE ANGLE
R6=R20/2;                       TOOL RADIUS CALCULATION
R7=3;                           APPROACH DISTANCE
R10=200;                        FEEDRATE
R1=R1+R6;                       CIRCLE RADIUS REDEFINITION
G00 X=((R1+R7)*COS(R3)) Y=((R1+R7)*SIN(R3))
Z2
G01 Z-5 F150
START: G01 X=(R1*COS(R3)) Y=(R1*SIN(R3)) F=R10
R3=R3+R5
IF R4>R3 GOTOB START
G01 X=(R1*COS(R4)) Y=(R1*SIN(R4))
X=((R1+R7)*COS(R4)) Y=((R1+R7)*SIN(R4))
G53 G00 Z-110 D0
M30
```



Example 5:

Elaborate a parametric program in order to execute a ellipse to 360 degrees.

```
%_N_EXE_5_MPF
;$PATH=/_N_WKS_DIR/_N_EXAMPLE_WPD
G17 G64 G71 G90 G94
G53 G0 Z-110 D0
T18;_____
M6
G54 D1 S3500 M3
R1=80;                                  LARGER AXIS LENGTH
R2=50;                                  SMALLER AXIS LENGTH
R20=10;                                 TOOL DIAMETER
R1=((R1+R20)/2) R2=((R2+R20)/2);        ELLIPSE  RADIUS CALCULATION
R3=0;                                   START ANGLE
R4=360;                                 END ANGLE
R5=1;                                   INCREASE ANGLE
R7=3;                                   APPROACH DISTANCE
R10=200;                                FEEDRATE
G00 X=((R1+R7)*COS(R3)) Y=((R2+R7)*SIN(R3))
Z2
G01 Z-5 F150
START: G01 X=(R1*COS(R3)) Y=-(R2*SIN(R3)) F=R10
R3=R3+R5
IF R4>R3 GOTOB START
G01 X=(R1*COS(R4)) Y=(R2*SIN(R4))
X=((R1+R7)*COS(R4)) Y=((R2+R7)*SIN(R4))
G53 G00 Z-110 D0
M30
```

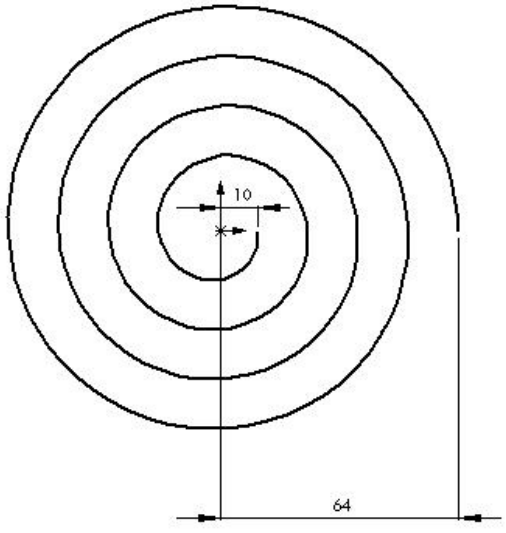


Example 6:

Elaborate a parametric program in order to execute a spiral:

```
%_N_EXE_6_MPF
;$PATH=/_N_WKS_DIR/_N_EXAMPLE_WPD
G17 G64 G71 G90 G94
G53 G0 Z-110 D0
T12
M6
G54 D1 S2250 M3
R1=10;                  START RADIUS
R20=64;                 END RADIUS
R3=0;                   START ANGLE
R7=5;                   NUMBER OF SPIRALS
R4=R7*360;              END ANGLE
R5=1;                   INCREASE ANGLE
R8=(R20-R1)/R4;         INCR. RADIUS_X_UNIT ANGLE.
R9=R8*R5;               INCR. RADIUS P/ INCREASE  ANGLE
R10=200;                FEEDRATE
G00 X=(R1*COS(R3)) Y=(R1*SIN(R3))
Z2
G01 Z-5 F150
START: G01 X=(R1*COS(R3)) Y=(R1*SIN(R3)) F=R10
R3=R3+R5 R1=R1+R9
IF R4>R3 GOTOB START
G01 X=(R20*COS(R4)) Y=(R20*SIN(R4))
G53 G00 Z-110 D0
M30
```

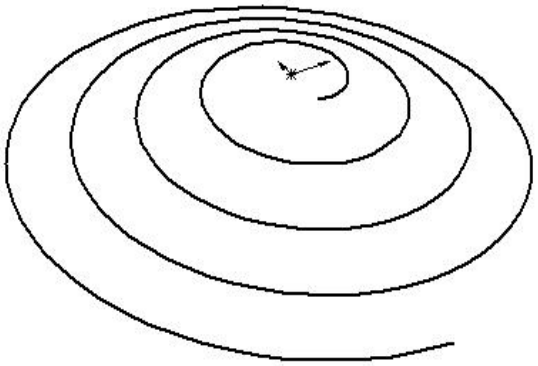
Start radius:	10mm
End radius:	64mm
Nº.espirals:	5
Start angle :	0 grau



Example 7:

Elaborate a parametric program in order to execute a spiral with a "Z" axis move.

Start angle:	10mm
End radius:	64mm
N.espirals:	5
Start angle:	0 graus
Start position"Z":	0
End position"Z" :	25

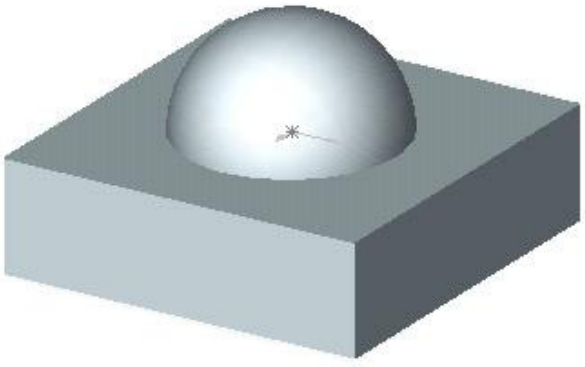


```
%_N_EXE_7_MPF
;$PATH=/_N_WKS_DIR/_N_EXAMPLE_WPD
G17 G64 G71 G90 G94
G53 G0 Z-110. D0T1;_____ M6
G54 D1 S2500 M3
R1=10;                  START RADIUS
R20=64;                 END RADIUS
R3=0;                   START ANGLE
R7=5;                   NUMBER OF ESPIRALS
R4=R7*360;              END ANGLE
R5=1;                   INCREASE ANGLE
R8=(R20-R1)/R4;         INCR. RADIUS_X_UNIT ANGLE.
R9=R8*R5;               INCR. RADIUS P/ INCREASE  ANGLE
R10=200;                FEEDRATE
R11=0;                  "Z" AXIS START POSITION
R12=25;                 "Z" AXIS END POSITION
R13=R12-R11;            DEPTH
R14=R13/R4;             INCREASE "Z" P/ UNIT. ANGLE
R15=R14*R5;             INCREASE "Z" P/ INCREASE ANGLE
G00 X=(R1*COS(R3)) Y=(R1*SIN(R3))
Z2
G01 Z=R11 F150
START:G01 X=(R1*COS(R3)) Y=(R1*SIN(R3)) Z=R11 F=R10
R3=R3+R5 R1=R1+R9 R11=R11-R15
IF R4>R3 GOTOB START
G01 X=(R20*COS(R4)) Y=(R20*SIN(R4)) Z=(-R12)
G53 G00 Z-110 D0
M30
```

Example 8:

Elaborate a parametric program in order to execute a sphere using a ball noose tool.

```
%_N_EXE_8_MPF
;$PATH=/_N_WKS_DIR/_N_EXAMPLE_WPD
G17 G64 G71 G90 G94
G53 G0 Z-110. D0
T1;
M6
G54 D1 S2500 M3
R1=50;                  RADIUS EPHERE
R20=10;                 TOOL DIAMETER
R20=R20/2;              TOOL RADIUS
R1=R1+R20;              TOOLS WAY - DEFINITION
R3=0;                   START ANGLE
R4=90;                  END ANGLE
R5=1;                   INCREASE ANGLE
R6=0;                   "Z" START POSITION
R7=5;                   APPROACH DISTANCE
R10=200;                FEEDRATE
CFTCP
G00 X=(R1+R7)*COS(R3) Y0
Z=R20+2
G01 Z=R6 F150
START: G01 X=R1*COS(R3) Y0 Z=R1*SIN(R3) F=R10
G02 X=R1*COS(R3) Y0 I=AC(0) J=AC(0)
R3=R3+R5
IF R4>R3 GOTOB START
G53 G00 Z-110 D0
M30
```



Example 9:

Elaborate a parametric program in order to execute a circular pocket:

```
%_N_EXE_9_MPF_DIR
;$PATH=/_N_WKS_DIR/_N_EXAMPLE_WPD
G17 G64 G71 G90 G94
G53 G0 Z-110. D0
T1;
M6
G54 D1 S2500 M3
R1=5;          “Z” RETURN
R2=0;          “Z” START POINT
R3=2;          SAFE DISTANCE
R4=-10;        “Z” END POINT
R5=20;         RADIUS POCKET
R6=75;         “X” CENTER POINT
R7=50;         “Y” CENTER POINT
R8=3;          INFEED DEPTH
R9=350;        FEEDRATE “X”
R10=175;       FEEDRATE “Z”
R11=5;         TOOLS RADIUS
R12=7;         SIDE INCREASE
R13=R2+R3;   CALCULOS - “Z” START POINT
IF (R11>R5) GOTOF ERROR
IF (R11=R5) GOTOF DRILL
IF (R11<R5) GOTOF STA_POC
ERROR:
MSG(“ERROR! TOOLS RADIUS  > POCKET RADIUS”)
M0
GOTOF END_PROG
DRILL:
G0 Z=R1
X=R6 Y=R7
Z=R13
G1 Z=R4 F=R10
GOTO END_PROG
STA_POC:
R20=R2-R8
R21=R6+R12
R22=R6+R5-R11
G0 Z=R1
X=R6 Y=R7
Z=R13
IF (R21<R22) GOTOF POC_2
POC_1:
IF R20<R4 GOTOF END_POC1
```

```
G1 Z=R20 F=R10
G1 X=R22 F=R9
G2 I=AC(R6) J=AC(R7)
G1 X=R6 Y=R7
R20=R20-R8
GOTOB POC_1
END_POC1:
G1 Z=R4 F=R10
G1 X=R22 F=R9
G2 I=AC(R6) J=AC(R7)
G1 X=R6
POC_2:
IF R20<=R4 GOTOF END_POC2
G1 Z=R20 F=R10
EXP:
G1 X=R21 F=R9
G2 I=AC(R6) J=AC(R7)
R21=R21+R12
IF (R21<R22) GOTOB EXP G1 X=R22
G2 I=AC(R6) J=AC(R7)
G1 X=R6
END_EXP:
R21=R6+R12
R20=R20-R8
GOTOB POC_2
END_POC2:
G1 Z=R4 F=R10
REPEAT ALARG END_EXP
END_PROG:
G0 Z=R1
G53 G0 Z-110 D0
M30
```

Example 10:

Elaborate a parametric program in order to execute a circular pocket using a 4th. axis .

```
 %_N_EXE_10_MPF
;$PATH=/_N_WKS_DIR/_N_EXAMPLE_WPD
G17 G71 G90 G94
G53 G0 Z-110 D0
T1
G54 D1 S3500 M3
R1=100;              PART DIAMETER
R2=40;               RADIUS POCKET
R13=5;               RETURN POINT
R3=0;                REFERENCE POINT
R4=2;                SAFE DISTANCCE
R5=-15;              DEPTH (ABS)
R6=80;               “X” CENTER POINT
R7=0;                START ANGLE “W - AXIS”
R8=2;                INFEED DEPTH
R9=8;                RADIUS INCREASE
R10=500;             FEEEDRATE  (X,W)
R11=250;             DEPTH FEEEDRATE (Z)
R14=16;              TOOLS RADIUS
R15=0;               START ANGLE
R16=1;               INCREASE ANGLE
R50=R1*3.14159;      PART PERIMETER
R51=R3+R4-R8
R53=R9
R54=R15
G64 G0 X=R6 Y0 W=R7
Z=R13
Z=R3+R4
N60 FGROUP (X,W)
N70 FGREF[W]=((R1-R5)/2)
START: G1 Z=R51 F=R11
START2:
IF R14>R2 GOTOF END
AAA: IF (R14+R9)>R2 GOTOF ACAB
RADIUS: G1 X=(R6+(R9*COS(R15))) W=(R7+(R9*SIN(R15)))*360/(R1*3.141)
R15=R15+R16
IF R15<360 GOTOB RADIUS R15=R54
R9=R9+R53
GOTOB AAA
FINIS: G1 X=R6+((R2-R14)*COS(R15)) W=(R7+(((R2-R14)*360/R50))*SIN(R15))
N430 R15=R15+1
N440 IF R15<360 GOTOB RADIUS
END2:
```

```
END:R51=R51-R8
IF R51>R5 GOTOB START
G1 Z=R5 F=R11
REPEAT START2 END2
G53 G0 Z-110 D0
M30
```

PART II - OPERATION

1 - COMMAND PANEL - CNC SIEMENS 840

The Command Panel is useful to display data, and to program, operate and run command functions.Therefore, it is divided into four additional panels::

- Display Panel;
- Programming Panel;
- Work Mode Panel
- Remote Panel.

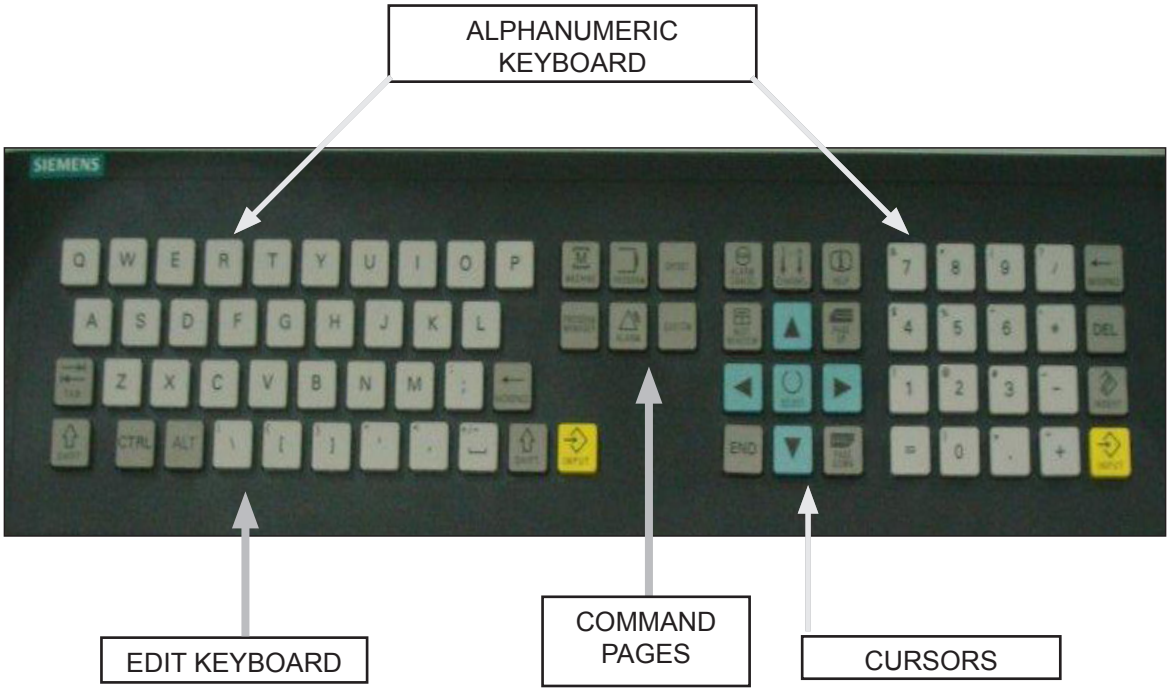
COMMAND PANEL VIEW

1.1 - DISPLAY PANEL

a) Display Panel Details

1.2 - PROGRAMMING PANEL

a) Programming Panel Details



b) Programming Panel Description

NAME	DESCRIPTION
CURSORS	**-** ↑ ,↓,→,←**:** Move the cursor up, down, right and left. **- *PAGE UP:*** Change the page up. **- *PAGE DOWN:*** Change the page down. **- *NEXT WINDOW:*** Change the window at the same page. **- *END:*** Let the cursor at the and of block.

NAME	DESCRIPTION
COMMAND PAGES	This keys give access into the main pages. There are: ***- MACHINE:*** Shows the machine coordinate, works coordinate and relative coordinate. ***- PROGRAM MANEGER :*** Shows the files manager. ***- PROGRAM :*** Shows the edit program pages . ***- OFFSET :*** *Shows the tools preset, work piece reference definition, customer parameters, customer variables, etc...* ***- ALARM:*** Shows the alarms and messages page. ***- CUSTOM:*** No fucntion ***- MENU SELECT:*** Shows the softkeys who gives access to other pages. ***- MENU FUNCTION:*** No fuction. ***- MENU USER:*** No fuction. ***- ALARM CANCEL:*** Cancels alarms and messages ***- GROUP CHANNEL:*** no function ***- HELP:*** Shows help page.
EDITION KEYS	***- INPUT:*** This key confirm the data input and starts a new block edition. ***- DEL:*** Delete data forward the cursor during the edition. ***- BACKSPACE:*** Delete data behind the cursor during the edition.. ***- INSERT:*** Rewrite the data.
ALPHANUMERIC KEYBOARD	Letters, numbers and others characterses to insert and changes the data. ***- SHIFT:*** Useful to insert the secondary characterses. ***- CTRL:*** Useful to do some specials edit procedures. ***- ALT:*** Useful to insert some epecials characterses. ***- TAB:*** Useful to change the blanks.

1.3 - OPERATION PANEL

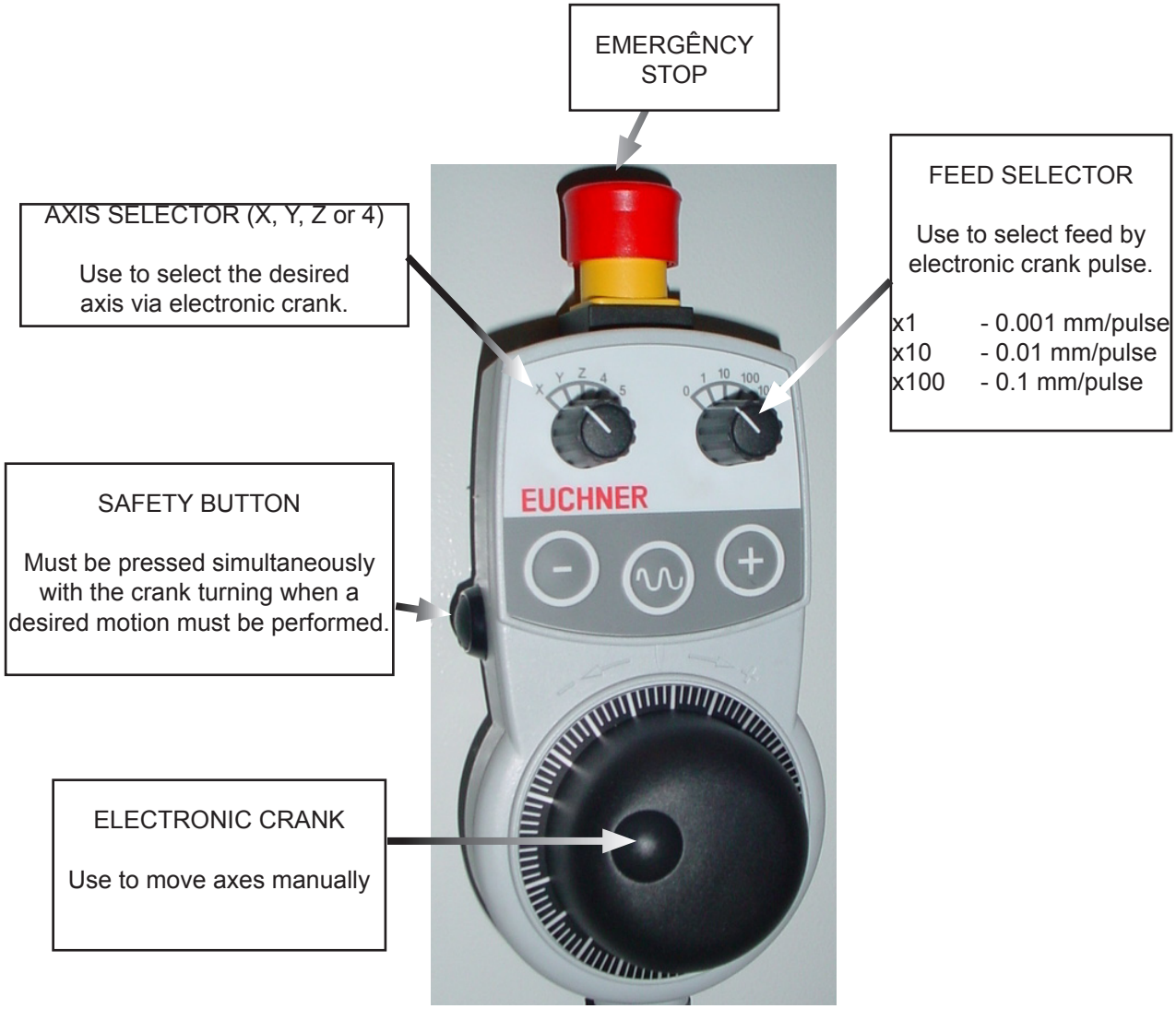
b) Operation Panel Description

NAME	DESCRIPTION
AUTO	Automatic execution mode.
BLOCK DELET	Activates /deactivates omission of program blocks starting with "/" (slash) during its run.
CHIP CONV. CCW	Turn on chip conveyor (counterclockwise - inverse).
CHIP CONV. CW	Turn on chip conveyor (counterclockwise - inverse).
CHIP CONV. STOP	Turn off chip conveyor.
CLTN AUTO	Turn on / off coolant via M08 and M09 function programming, respectively.
CLNT OFF	Turn off coolant manually.
CLNT ON	Turn on coolant manually.
CNC ON	Turn on command (CNC).
CYCLE START	Starts run the cycle.
CYCLE STOP	Stops run the cycle.
DRY RUN	Activates / deactivates the program test with fast axis motion.
EMERGENCY STOP	Emergency stop.
ENABLE REMOTE PANEL	Manual axis motion mode.
FEEDRATE OVERRIDE	Increases / decreases programmed axis feed during program run (0 to 200%)
FEED START	Enable axis feedrate
FEED STOP	Desable axis feedrate
HOME ATC	Reference the magazine
INDEX ATC	Indexes the tool magazine when the JOG mode is active
JOG	Continuous axis motion mode
LOCK	Enable, desable program edit
MACHINE ON	Turn on machine.
MDA	Manual data entry.
OK OPERATOR	Operator's confirmation key for messages displayed on cnc screen.

NAME	DESCRIPTION
OPEN CLOSE DOOR	Enable, desable open close door.
RAPID	Increase the feedrate test.
REPOS	Restart in the middle of the program.
REF. POINT	Useful to do axis reference.
RESET	Cancels, Reset the cycle / alarms.
SETUP	Enable, desable work with the door opened.
SINGL BLOCK	Activates / deactivates block by block program execution.
SPIDLE DEC	Reduces 10% in the revolution.
SPIDLE DEC	Increases 10% in the revolution.
SPINDLE LEFT	Indicates whether the arbor shaft is turning in counterclockwise direction.
SPDL RIGTH	Indicates whether the arbor shaft is turning in clockwise direction.
SPDL STOP	Indicates whether the arbor shaft is stopped.
TEACH IN	Program preparation via manual positioning.
WASH GUN	Enable wash gun.
WCS / MCS	Change the coordinate system: Machine coordinate system and Work coordinate system.
X,Y,Z	Select axis via JOG.
4,5,6	Select optional axis.
	Select feed by handwheel.
100%	Works with the feedrate in 100%
+	Move the axis in the positive direction.
-	Move the axis in the negative direction.

1.4 - REMOTE PANEL :

Remote Panel is useful to move machine axes manually.


EMERGÊNCY STOP
AXIS SELECTOR (X, Y, Z or 4)
Use to select the desired axis via electronic crank.
FEED SELECTOR
Use to select feed by electronic crank pulse.
x1 - 0.001 mm/pulse
x10 - 0.01 mm/pulse
x100 - 0.1 mm/pulse
SAFETY BUTTON
Must be pressed simultaneously with the crank turning when a desired motion must be performed.
ELECTRONIC CRANK
Use to move axes manually
EUCHNER

2 - INITIAL OPERATIONS

2.1 - TURN MACHINE ON

- Turn master switch on (located on the rear side of machine)
- Press ***"CNC ON"*** button to turn CNC on (wait for the initialization process)
- Deactivate emergency buttons (***"EMERGENCY STOP"*** on command and remote panels)
- Open and close the door
- Press ***"OK OPERATOR"*** key
- Press ***"HOME ATC"*** key, and wait the ATC reference.
- Press ***"RESET"*** key.

2.2 - REFERENCE MACHINE AXES:

Before axis referencing, check whether or not they are already close to the reference point. If yes, move them towards negative direction (X-, Y- and Z-) so to move them away from this position, as per chapter 3.

To reference axes, proceed as follows:

2.2.1 - Reference axis :

- Close the door.
- Press ***"JOG"***
- Press ***"REF POINT"***
- Press ***"HOME"***
- Press ***"CYCLE START"***

NOTES:
- "HOME" key remains lit while referencing it's in progress;
- At end, "JOG" key lights on automatically. Indicating that referencing has finished;;
- Do not move the machine while referencing has not been finished.

2.2.1 - Reference axis by axis :

- Close the door.
- Press ***"JOG"***
- Press ***"REF POINT"***
- Press the key corresponding to the axis to reference (**X, Y, Z** or **W).**

2.2.2 - TURN MACHINE OFF

- Press emergency button (***"EMERGENCY STOP"***)
- Turn master switch off (located on the rear side of machine)

2.3 - MOVE AXES MANUALLY

2.3.1 - Move axes via Continuous Jog (1)

– Press **"M MACHINE"**.
– Press **"JOG"**.
– Press ***"POS"*** key to view positions.
– Press the key corresponding to the axis (***X, Y, Z,*** or ***W***).
– Press **"JOG".**
– Press ***"POS"*** key to view positions.
– Hold pressed the key corresponding to displacement direction "**+**" or "**-**".
– Press ***"POS"*** key to view positions.
– The "JOG" feed may be adjusted via ***"OVERRIDE"*** selector.
– For rapid traverse press the desired axis button and the **"RAPID"** key simultaneously.

NOTE: *To move the axis it's required to keep the* ***"SETUP"*** *switch in unlocked position.*

The "W" axis, or 4th axis, can be enable just when the key "WCS MCS" of operator's panel be desable.
When de OVERRIDE stay in 100%, the feedrate axis move it's 2000mm/min. To change this feedrate it's necessary change the OVERRIDE or :

– Press **"JOG"**.
– Press **"M MACHINE"**
– Press **[>]** softkey **(**if necessary).
– Press **[*SETTINGS*]** softkey.
– Put the cursor in the blank **"set-up feedrate G94"** and type the new feed value. Ex: "***3000***"
– Press the **"*INPUT*"** key.

To back the standard value (2000 mm/min) it's necessary type "0" at the blank **"set-up feedrate G94"***.*

2.3.2 - Move axes via Continuous Jog (2 - remote panel).

– Press **"M MACHINE"**.
– Press **"JOG"**.
– Press **"ENBLE REMOTE PANEL"**.
– On remote panel, select the axis to be moved, **"X", "Y", "Z"** or **"4"** (opc).
– Press the key corresponding to displacement direction "**+**" or "**-**",holding the safety button pressed (located on the remote panel side).
– For rapid traverse press the desired axis button and **"RAPID"** simultaneously.

NOTE: *Turn "****SETUP****" switch to operate with the door open.*

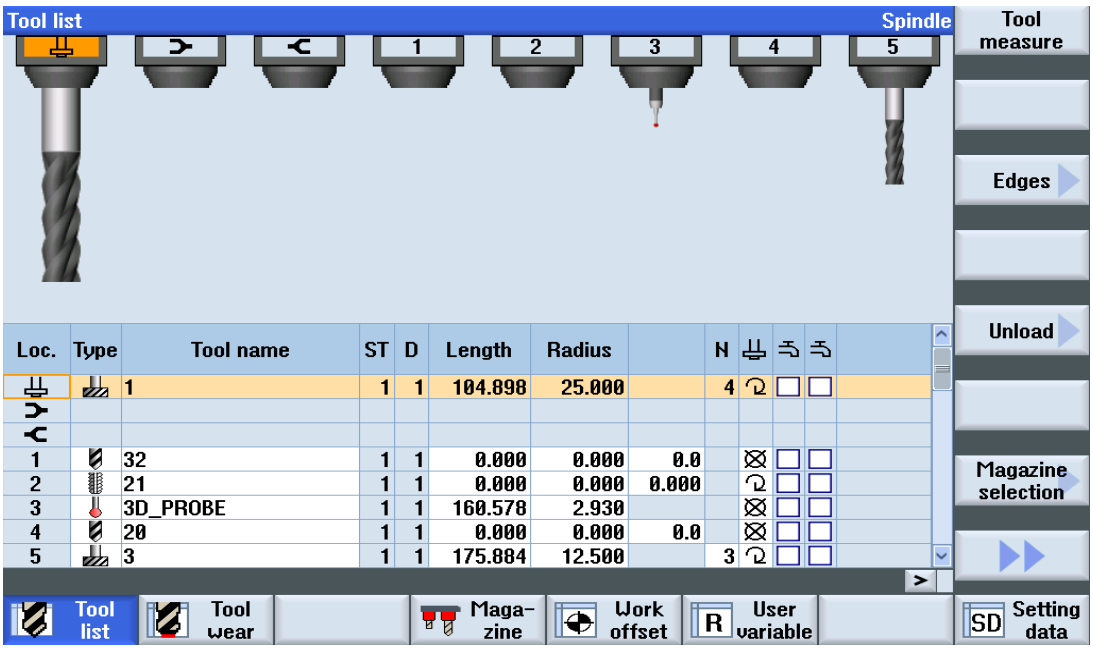
2.3.3 - Move axes via electronic crank

- Press **“M MACHINE”**.
- Press **“JOG”**.
- Press **“ENABLE REMOTE PANEL”**.
- On remote panel, select the axis to be moved, **“X”, “Y”, “Z”** or **“4“** (opc).
- On remote panel, select feed by pulsation (**1**, **10** or **100**)
- Turn the crank while holding the safety button pressed (located on the remote panel side).

***OBSERVAÇÃO**: To run another operation, it is necessary to position feed by pulsation in “**0**” and press “**ENABLE REMOTE PANEL**” key again to deactivate it..*

2.3.4 - MOVE AXES VIA INCREMENTAL JOG

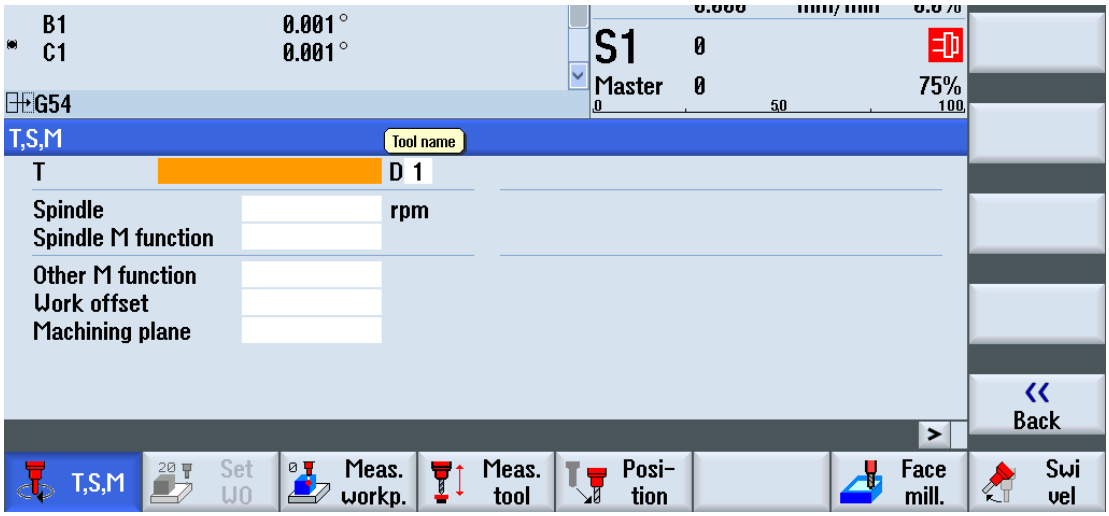
- Press “**JOG**”.
- Press “**M MACHINE**”
- Acionar a softkey ***[SETTING]***
- Place the cursor on “INCREASES” and type the increment value.Ex.:**1000**
- Press **“INPUT”**.
- Press**“VAR”**.
- Press the key, on the operation panel, corresponding to the axis and displacement direction (**X**, **Y**, **Z**, **W**).
- Press the key, on the operation panel corresponding to displacement direction “**+**” or “**-**”



2.4 - FUNCTION T,S,M.

The function **"T,S,M"** it's useful to execute some basic operations, normally during the machine setup. To access this function it's necessary:

– Press **"JOG"**.
– Press **"M MACHINE"**
– Press **[T,S,M]** softkey.



Following will be described some possible operations with the **"T,S,M"** function.

2.4.1 - Change tools.

After press the **"T,S,M"**, function it's necessary:

– Place the cursor at the blank **"T"**.
– Type the tool number desired. Example: 1 (T01)
– Press **"INPUT"**
– Se necessário, Place the cursor at the blank **"*D*"** (if necessary) and type the geometry tool number desired. Example: **1** (D1)
– Press **"INPUT"**
– Press **"CYCLE START"**

***NOTE**: To select the desired tool at the page "Tool List" it's necessary:*

– Press ***[SELECT TOOL]*** softkey.
– Place the cursor at the desired tool.
– Presse ***[IN MANUAL]*** softkey.

2.4.2 - Turn the spindle ON.

After press the **"T,S,M"**, function it's necessary:

- Place the cursor at the blank **"SPIN"** and type the RPM value. Example: "**2000**" (2000 RPM)
- Press **"INPUT"**
- Place the cursor at the blank **"*Spindle Function M*"** and select the spindle direction through the **"SELECT"** .
- Press the **"CYCLE START"**

2.4.3 - SPINDLE ORIENTATION.

After press the **"T,S,M"**, function it's necessary:

- Place the cursor at the blank **"*Spindle Function M*"** and select the spindle orientation function.
- Place the cursor at the blank **"*Spindle Orientation Position*"** and type the angle position. Example: 45.
- Press **"INPUT"**
- Press **"CYCLE START"**

2.4.4 - Miscellaneous functions

After press the **"T,S,M"**, function it's necessary:

- Place the cursor at the blank **"***Others fuctions "M"*** and type the "M" fuction desired. Example: **8**
- Press **"INPUT"**
- Press **"CYCLE START"**

2.4.5 - WORK COORDINATE SYSTEM (G54 - G57 ou G505 - G599).

After press the **"T,S,M"**, function it's necessary:

- Place the cursor at the blank *"Work coordinate"* and type the workpiece reference desired. Ex.: **1** (G54), or press the **"SELECT"** key.
- Press **"INPUT"**
- Press **"CYCLE START"**

***NOTES**: To show the workpiece reference coordinate system, it's necessary to activate the "**MCS WCS**" key.*

It's possible Select the workpiece reference through the "WORK PIECE REFERENCE" page. To do this it's necessary:

- Press the **[select work off.]** softkey
- Place the cursor at the workpiece reference desired.
- Press the **[in manual]** softkey.

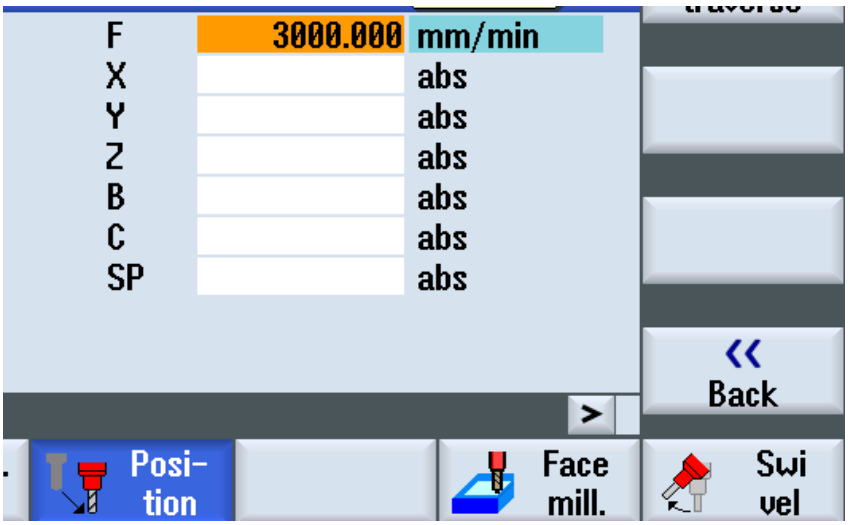
2.4.6 - Measurement unit reference

After press the **"T,S,M"**, function it's necessary:

- Place the cursor at the blank **"***Measurement unit***"** and press **"SELECT"**, to change in **"mm"** or **"IN"** .
- Press **"CYCLE START"**

2.5 - RAPID TRAVERSE POSITIONING THROUGH THE "POSITION" FUNCTION

The function "Position" it's useful to do positioning according to workpiece reference activated. We use this function to try the workpiece reference place.



To access this function it's necessary:

- Press the **"JOG"** key.
- Press the "**MACHINE**" key.
- Press the **[>]** softkey (if necessary).
- Press the **[Position]** softkey (if necessary).
- Place the cursor at the blank **"** *F* **"** and type the feedrate value or press the **[*FAST MOVE*]** softkey to do the positioning in G0.
- Press **"INPUT"**
- Place the cursor at the blank **"***X***"** and type the coordinate to be reached.

Example: **-15**

– Acionar a tecla **"INPUT"**.
– Place the cursor at the blank **"***Y***"** and type the coordinate to be reached. Example: **-20**
– Press **"INPUT"**
– Place the cursor at the blank **"***Z***"** and type the coordinate to be reached. Example: **10**
– Press **"INPUT"**
– Press **"CYCLE START"**

NOTES: *Using the* ***"SELECT"*** *key, it's possible change the coordiante between absolute or incremental.*

Using the ***"SELECT"*** *tool it's possible choose the feedrate between* **"mm/min."** *or* **"mm/rot.".**

2.6 - LOAD THE TOOLS AT THE MAGAZINE (ATC) THROUGH THE SPINDLE.

– Press **"JOG"**
– Press **"MACHINE"**
– Through the function **"T,S,M"**, change the first tool to be loaded.
– Press **"CHANGE TOOL"** and insert the tool at the spindle.
– Through the function**"T,S,M"**,change the first tool to be loaded.
– Press **"CHANGE TOOL"** and insert the tool at the spindle.
– Repeat the same procedure to the next tool desired.

3 - MANUAL DATA INPUT (MDA)

"MDA" mode it's applied to run simple operations, such as, change tool, turn Spindle on, move axes to any position, etc.

It enables the creation of a program containing up to 10 blocks, which it's edited and runs in the same format as a normal program.

To work with "MDA" mode, proceed as follows

- Press ***"MDA".***
- Press **"MACHINE"**.
- Enter the required instructions. Ex: **S800 M3**; (turns spindle on at 800 RPM).
- Press ***"INPUT".***
- Press ***"CYCLE START".***

> ***NOTES:***
> *To delete a program edited in MDA, press **[DELETE BLOCKS]** softkey;*

Example - Change tools via MDA:

- Press ***"MDA".***
- Press **"MACHINE"**.
- Enter "T" and the desired tool number. Example: T01
- Press ***"INPUT".***
- Enter: M6
- Press ***"INPUT".***
- Press **"CYCLE START"** (until completing the "tool change")

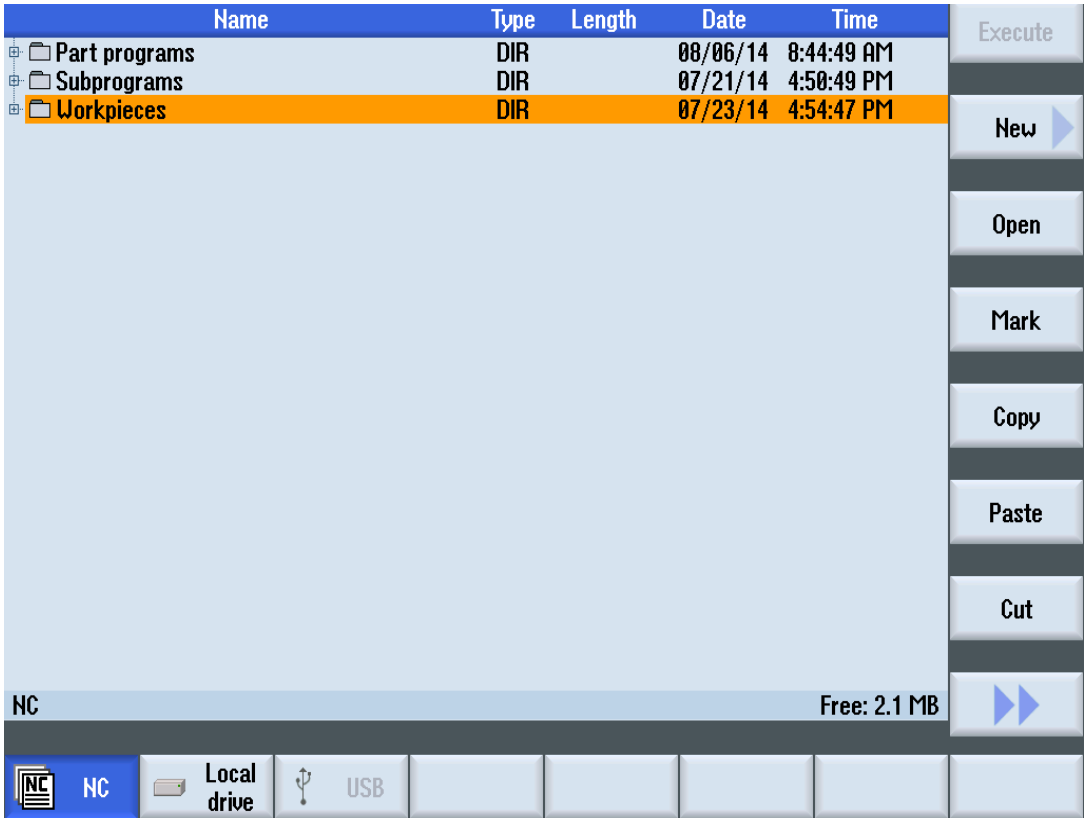
> ***NOTES:*** *To work with MDA mode it's necessary turn "SETUP" switch to operate with the door closed.*

4 - PROGRAM EDIT

In the Siemens 828 Control, it's possible the user to access the program situated in machine memory (NC memory), Compact flash, USB Card and Ethernet.

In the NC memory the edition can be done by three different directory:

- Parts programs
- Sub programs
- Works pieces



In the "WORK PARTS" directory, it's possible the user to create other subdirectorys to supply the programs.

4.1 - CREATE A NEW DIRECTORY

- Press **"PROGRAM MANAGER"**.
- Press **[*NC*]** softkey.
- Use cursor keys (◄, ►, ▲, ▼), to place the cursor on **"***WORK PIECE REFERENCE***"**.
- Press **"INPUT"**.
- Press **[*NEW*]** softkey.
- Press **[*DIRECTORY*]**.
- Enter Directory name. Ex: **"TST1"**
- Press **[*OK*]** softkey.

4.2 - CREATE A NEW PROGRAM

- Press **"PROGRAM MANAGER"**.
- Press **[*NC*]** softkey.
- Use cursor keys (◄, ►, ▲, ▼), to place the cursor on directory where the program will be create.
- Press **"INPUT"**.
- Press **[*NEW*]** softkey.
- Press **[*programGUIDE G code*]** or ***[ShopMill]*** softkey.
- Enter Program name. Ex: **"PROG1"**
- Press **[*OK*]** softkey.

4.3- RENAMING A PROGRAM

- Use cursor keys (▲, ▼) to place the cursor on the program to be renamed.
- Press ***[►►].***
- Press ***[PROPARTIES] sofkteys.***
- Enter new program name. Ex: **PROG2.MPF**
- Press **[*OK*]** softkey.

4.4 - COPYING PROGRAM.

- Press ***"PROGRAM MANAGER".***
- Press ***[NC]*** softkey.
- Use cursor keys (◄, ►, ▲, ▼), to place the cursor on directory or program to copy
- Press ***[COPY].***
- Use cursor keys (◄, ►, ▲, ▼), to place the cursor on directory to paste
- Press ***[PASTE]***.

4.5 - DELETE A PROGRAM OR DIRECTORY

- Press ***"PROGRAM MANAGER".***
- Press***[NC]*** softkey.
- Use cursor keys (▲, ▼) to place the cursor on the program to be deleted.
- Press ***[DELETE]*** softkey***.***
- Press ***[OK]*** softkey***.***

NOTE: *This procedure must be applied with extreme caution, because when a programs deleted, there isn't way of restore it from the machine memory.*

4.6 - SELECT AN EXISTING PROGRAM IN THE DIRECTORY

- Press ***"PROGRAM MANAGER".***
- Press ***[NC]*** softkey.
- Use cursor keys (◄, ►, ▲, ▼), to place the cursor on directory or program to edit.
- Press **"INPUT"**

4.7 - INSERTING NUMBER OF BLOCKS AUTOMATICALLY.

Insert the number of block isn't obligatory to work with the machine, but, it's interesting insert the numbers in all blocks. Case the machine shown some alarm, it's easy to found the wrong block.

To insert numbers of block in edition page, it's necessary follows the next procedure:

- Press ***[►►]*** softkey.
- Press ***[RENUMBERING]*** softkey.
- Place the cursor at the blank *"**First block number**"* and type the number of first block of program. Ex: **10**
- Press the **"INPUT"** key.
- Place the cursor at the blank *"Increment"* and type the increase of number of blocks. Ex: **10**
- Press the **"INPUT"** key.
- Press the ***[OK]*** softkey.

4.8 - SEARCHING FOR DATA WITHIN A PROGRAM

a) Find a data via cursors (►,◄,▲ou▼.)

Press cursors until selecting the desired address, as explained below:

◄ - move the cursor left
► - move the cursor right
▲ - move the cursor up
▼ - move the cursor down

b) Direct search (direct to address)

- Press ***[SEARCH]*** softkey
- Type the data to be searched. Ex: **N10**
- Place the cursor at the blank "Direction" and select the option "Forward" or "Backward" through the **"SELECT"** softkey.
- Press the ***[OK]*** softkey.
- Place the cursor at the blank "Text" and type the information to be searched example: **N100**
- If necessary search again, press ***[CONTINUE SEARCH]*** softkey.

4.9 - COPYING DATA:

– Press ***[MARK]*** sofkey.
– Use cursor keys (◄, ►, ▲, ▼), to place the cursor on the block desired to copy.
– Press ***[COPY]*** softkey.
– Use cursor keys (◄, ►, ▲, ▼), to place the cursor on the block desired to paste.
– Press ***[PASTE]*** softkey.

NOTE: *After copy some data, it's possible paste it in another program.*

– Press ***"PROGRAM MANAGER".***
– Press ***[NC]*** softkey.
– Use cursor keys (◄, ►, ▲, ▼), to select the directory or program to paste.
– Press "**INPUT**"
– Use cursor keys (◄, ►, ▲, ▼), to place the cursor on the block desired to paste.
– Press ***[PASTE]*** softkey.

4.10 - DELETING PROGRAM DATA

– Use cursor keys (◄, ►, ▲, ▼) to place the cursor on the data desired to delete.
– Press **"DELETE".**

4.11 - DELETING PROGRAM BLOCKS:

– Use cursor keys (◄, ►, ▲, ▼) to place the cursor on the data block to delete.
– Press ***[MARK]*** key.
– Use cursor keys (◄, ►, ▲, ▼) to place the cursor on the block desired to delete.
– Press ***[CUT]*** softkey.

4.12 - PROGRAMMING CYCLES IN INTERACTIVE PAGE.

To programming this cycles using interactives pages:

– Place the cursor at the line desired.
– Press the softkey according to cycle desired. ex: ***[HOLE]*** and ***[THREAD]***
– Fill the blanks according to desired.
– Press ***[OK]***

4.13 - CHANGE SOME DATA IN THE INTERACTIVE PAGE.

– Place the cursor at the cycle to change.
– Press "►".
– Change the data.
– Press ***[OK]*** softkey.

5 - DATA COMMUNICATION

This chapter comprises the required DATA COMMUNICATION resources to handle, save, load, copy, backup, etc. All data resident in the machine intended for equipment operation.

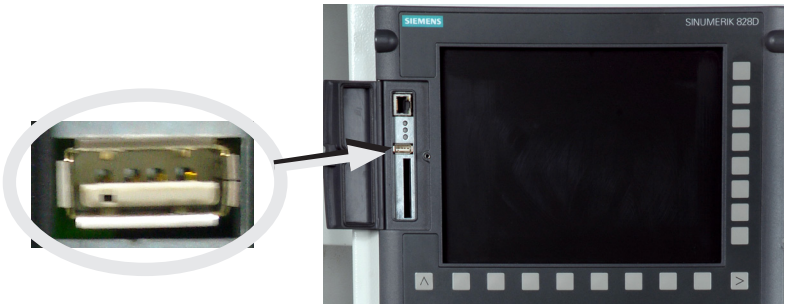
This data may be: machine parameters, programs, tool offsets, work coordinate offset, settings, “R” variables and global variables.

The mentioned actions require some elements for its execution, such as USB, DATA TRANSMISSION APPLICATIONS, OTHER PERIPHERICS, etc., whose instructions are the following.

5.1 - COMMUNICATION THROUGH USB ( )

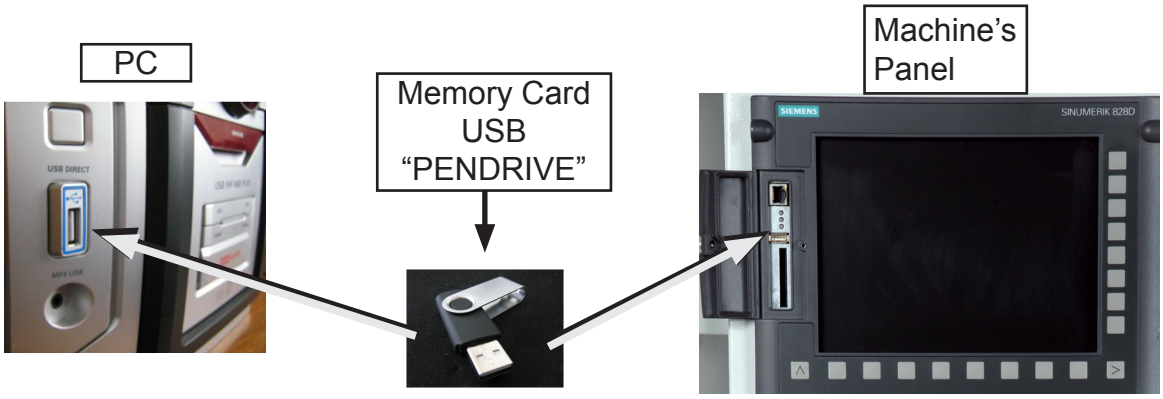
This port allows read and save same data through USB flash card - usually known “pen drive”. This card works with the system USB (Universal Serial Bus) to do the communication.

The “D Line” machines (with CNC Siemens 828) is provided with a USB port situated in the Panel in according to the figure below.



5.1.1 - Recommended hardware for reading and writing USB :

To read and write on USB flash card we recommend the interface USB 2.0 tip “A”.



5.1.2 - View files in the USB.

– Press **“PROGRAM MANAGER”.**
– Press ***[USB]*** softkey.

After this, will be shown all programs and directories storeds on USB. It’s possible do the same actions like an others memory cards. ex: create new directories, edit programs, delete programs, copy programs, paste programs, etc...

5.1.3 - Load a program from memory card (USB).

– Press **“PROGRAM MANAGER”.**
– Press **[*USB*]** softkey.
– Use cursor keys (◄ ►▲▼) to place the cursor on the program desired to copy.
– Press **[*COPY*]** softkey.
– Press ***[NC]*** softkey.
– Use cursor keys (◄ ►▲▼) to place the cursor on the directory desired to copy.
– Press ***[PASTE]*** softkey.

5.1.4 - Saving program from memory card.

– Press **“PROGRAM MANAGER”.**
– Press ***[NC]*** softkey.
– Use cursor keys (◄ ►▲▼) to place the cursor on the program desired to copy.
– Press **[*COPY*]** softkey.
– Press **[*USB*]** softkey.
– Use cursor keys (◄ ►▲▼) to place the cursor on the directory desired to copy.
– Press ***[PASTE]*** softkey.

5.2 - COMMUNICATION THROUGH ENTHERNET

This port allows the communication between the machine and the directory shared located in a PC. This kind of communication is called ETHERNET.

The “D Line” machines (with CNC Siemens 828) is provided with a “ETHERNET” port situated in the Panel, in according to the figure below.

5.2.1 - Recommended hardware for reading and writing ETHERNET :

Cable RJ-45. This cable, must be connected in a PC. This PC need to be a shared directory with the programs to be executed.

5.2.2 - View files in the PC.

- Press the **"PROGRAM MANAGER"** key.
- Press the ***[►►]*** softkey until ***[ETHERNET]*** is displayed.
- Press the ***[ETHERNET]*** softkey .

After this, will be shown all programs and directories storeds on ETHERNET. It's possible do the same actions like an others memory cards. ex: create new directories, edit programs, delete programs, copy programs, paste programs, etc...

5.2.3 - Load the files in the machine.

- Press **"PROGRAM MANAGER"**
- Press ***[►►]*** softkey until ***[ETHERNET]*** is displayed.
- Press ***[ETHERNET]*** softkey.
- Place the cursor at the program desired using the ►,◄,▲ and▼ keys.
- Press ***[COPY]*** softkey.
- Press ***[NC]*** softkey.
- Place the cursor on the directory desired using the ►,◄,▲and▼ keys.
- Press ***[PASTE]*** softkey.

5.2.4 - Saving program from PC.

- Press **"PROGRAM MANAGER"** .
- Press ***[►►]*** softkey until ***[NC]*** is displayed.
- Press **[NC]** softkey.
- Place the cursor at the directory or program desired using the ►,◄,▲and▼ keys.
- Press ***[COPY]*** softkey.
- Press ***[ETHERNET]*** softkey.
- Place the cursor at the directory using the ►,◄,▲and▼ keys.
- Press ***[PASTE]*** softkey.

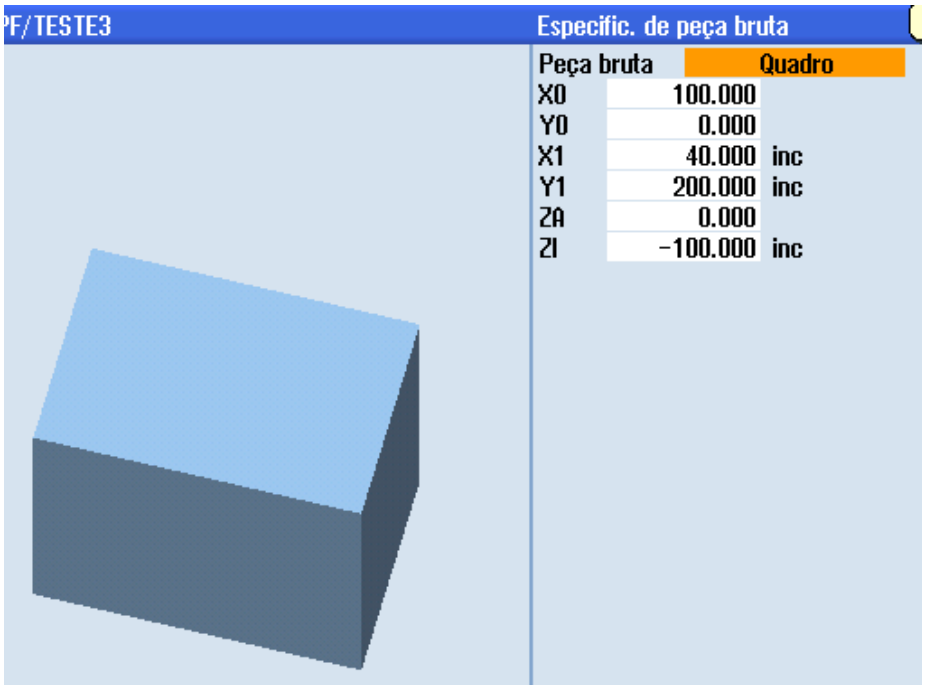
6 - PROGRAM TEST

Every programs must be tested before the execution. The “D line” presents many ways to do this tests.

To execute the graphical test it’s necessary insert some data in the begin of program, like the procedure below:

- Place the cursor in the begin of program to try.
- Press ***[Various]*** softkey.
- Press ***[Material]*** softkey**.**
- Place the cursor at the blank **“Material”,** and select the type of material using **“SELECT”**
- Fill the blanks according to material
- Press ***[OK]*** softkey.

Example of material in the graphic page.



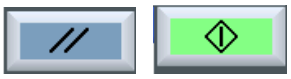
The material data is displayed in the program like the example below:

```
WORKPIECE(,,,"BOX",0,0,-25,-80,0,0,100,100)
G40 G64 G71 G90 G94
G53 G00 Z0 D0
T1;FRESA DE TOPO D10
M6
G54 D01 S2000 M3
```

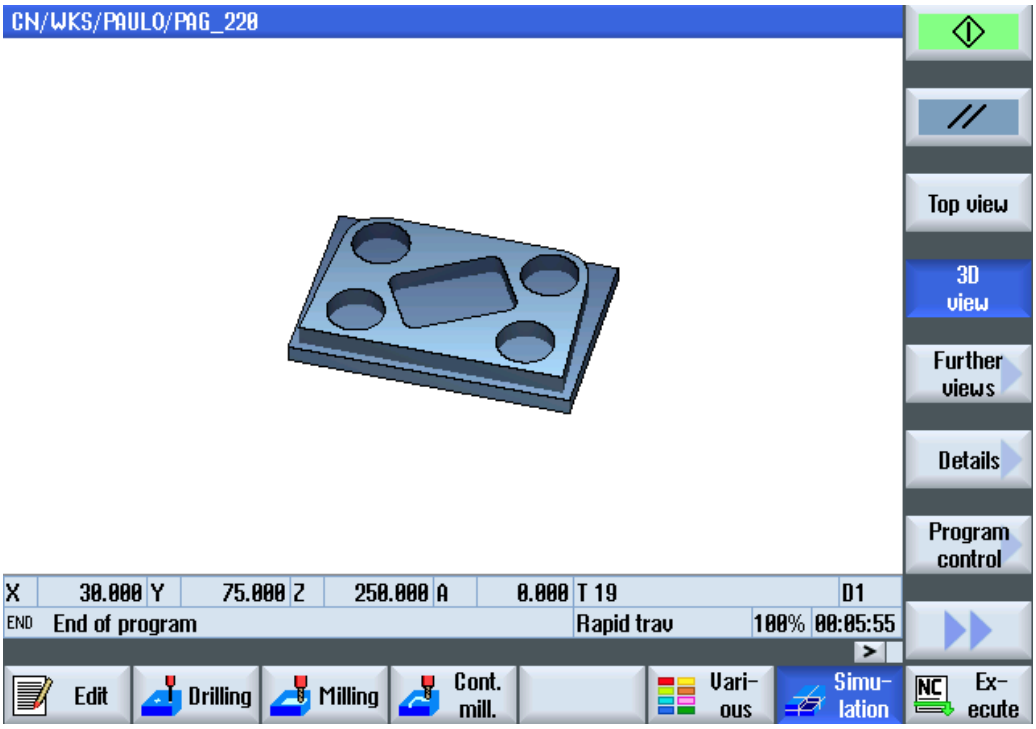
6.1 - EXECUTE GRAPHICAL TEST (WAY 1).

In the edition page:

- Press ***[SIMULATION]*** softkey.
- Wait some seconds.
- Press ***[//]*** softkey (reset).
- Press ***[START]*** softkey.



Example of graphical test:



The figure above shown the 3d graphical view. Others views can be displayeds in the graphical page. To change this views it's necessary select the view desired through the softkeys. Also, it's possible to see the way of tool, enlarge, reduce or turn the views.

To enlarge the view:

way 1:

- Press ***[DETAIL]*** softkey
- Place the cursor at the detail to be enlarged
- Press ***[ZOMM +]*** softkey to enlarge or ***[ZOMM -]*** to reduce.

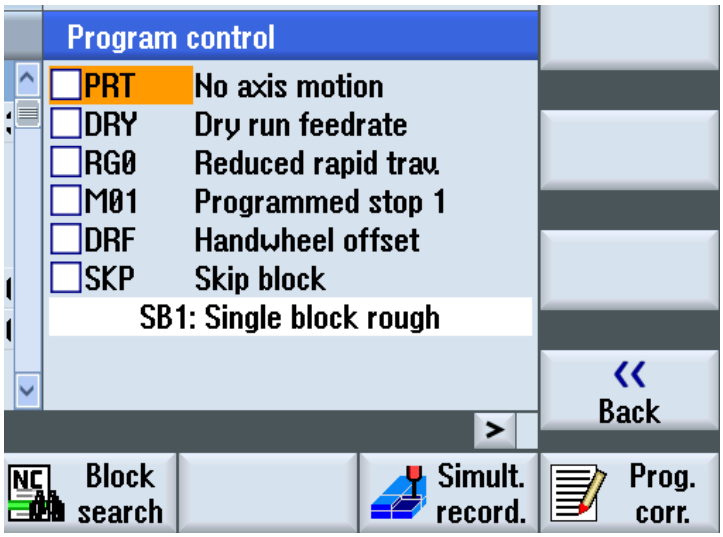
way 2:

- Press ***[DETAIL]*** softkey.
- Press ***[ZOOM ►]*** softkey.
- Place the cursor at the detail to be enlarged
- Press ***[ZOOM +]*** softkey to enlarge or ***[ZOOM -]*** to reduce.

6.2 - EXECUTE GRAPHICAL TEST (WAY 2).

The graphical test allows to see a better tool's movement. To this test it's necessary:

- Access the program to try
- Press ***[EXECUTE]*** softkey.
- Press ***"AUTO"***
- Press ***[►]*** softkey.
- Press ***[PROG. CONT.]*** softkey.
- Place the cursor in ***"PRT"*** .
- Press ***"SELECT"***
- Place the cursor in ***"DRY"***.
- Press ***"SELECT"*** key.
- Press ***[Back]*** softkey.
- Press ***[SIMULT. RECORD.]*** softkey.
- Press ***"CYCLE START"***



6.3 - TESTING PROGRAMS WITHOUT MOVING THE AXES

- Access the program to try
- Press ***[EXECUTE]*** softkey.
- Press ***"AUTO"***
- Press ***[►]*** softkey.
- Press ***[PROG. CONT.]*** softkey.
- Place the cursor in ***"PRT"*** .
- Press ***"SELECT"***
- Place the cursor in ***"DRY"***.
- Press ***"SELECT"***
- Press ***[Back]*** softkey.
- Press ***"CYCLE START"***

6.4- EXECUTING PROGRAM TEST IN DRY RUN MODE (RAPID FEED - DRY):

This test it's applied to check the workpiece reference, preset and sequence of motions that will be done during machining. In this test every moviments are executed with a rapid feedrate.

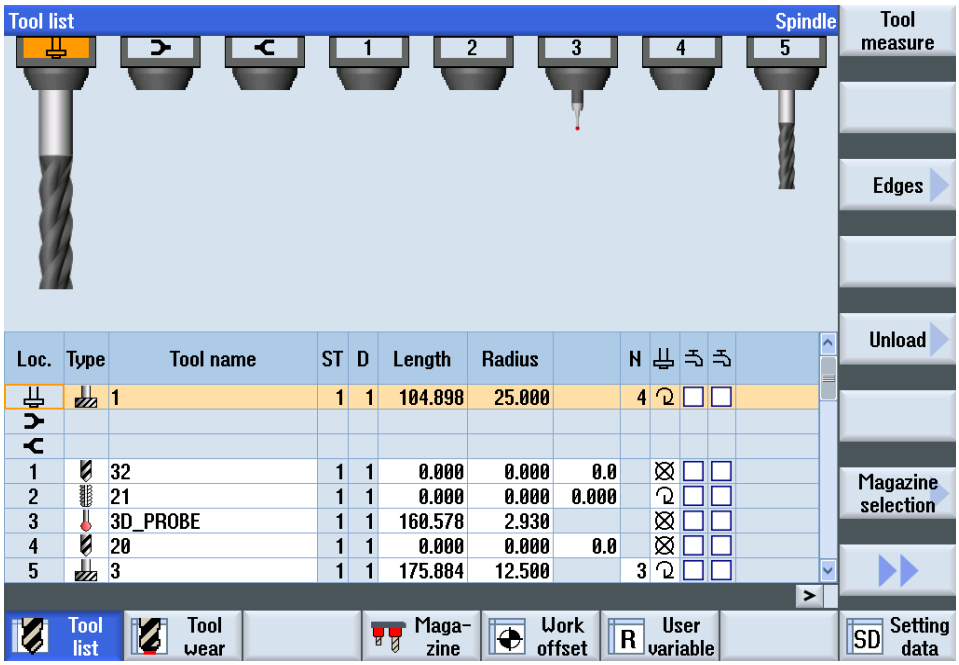
- Select the current program.
- Press ***[EXECUTE]*** softkey**.**
- Press ***"AUTO"***
- Hold ***"*****DRY RUN*****"*** during 2 seconds.
- Press ***"*****CYCLE START*****"***

NOTES*: Before execute this test is necessary do the tool's reference and the workpiece reference. In this case, the "Workpiece reference" in "Z" must be moved in a safe position above the part cause the machine will move the axis in a empty space.*
After the test i's necessary back the "Workpiece reference" in "Z" to original position.

7 - TOOL REFERENCE

Tool preset is a process that informs to machine the tool dimensions.

- Press **“OFFSET”**.
- Press **[*TOOL LIST*]** softkey.



7.1 - CREATE A NEW TOOL

This procedure is necessary just when unexpectedly some tool will be deleted. Therefore, this procedure isn’t necessary to start work in the machine if the tools was created before.

But, if necessary create a new tool, do the procedure below:

1º - Activate the tool T0 using the fuction **“T,S,M”***:*

- Press **“JOG”**
- Press **“M MACHINE”**
- Press **[T,S,M]** softkey.
- Place the cursor at the balnk **“T”**.
- Type **“0”**.
- Press **“INPUT”**
- Press **“CYCLE START”**

2º - Access the tool list page:

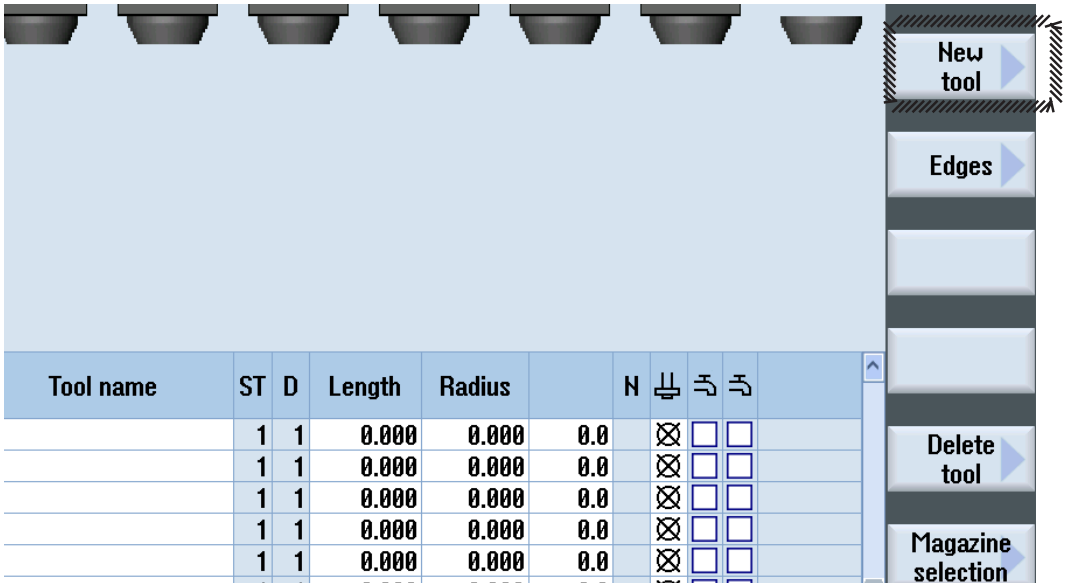
- Press **“MENU SELECT”**
- Press ***[SELECT TOOL]*** softkey.
- Press ***[TOOL LIST]*** softkey.

3º - Create a new tool:

- Place the cursor in the end of tool list, in a empty blank.
- Press **[New Tool]** softkey**.**
- Choose one of tool type possible using the softkeys.
- Use "***BACK SPACE***" to erase the name of the tool.
- Type the number desired. (This number must be different then other tools)
- Press **"INPUT"**

4º - Load the tool in the magazine:

- Place the cursor at the tool to be loaded
- Press **[*LOAD*]** softkey.
- Press **[*SPINDLE*]** softkey.



5º - Input the tool in the Spindle manually.

6º - Insert the tool in the magazine using the **"T,S,M"** *function:*

- Press **"JOG"**
- Press **" M MACHINE"**
- Press **[>]** softkey (if necessary)
- Press **[T,S,M]** sotkey.
- Place the cursor at the blank **"T"**.
- Type **"0"**.
- Press **"INPUT"**
- Press **"CYCLE START"**

7.2 - DELETE TOOL.

To delet tool is necessary:

1º - Using the function **"T,S,M"** *load the tool (to be deleted) in the spindle:*

- Press **"JOG"**
- Press **"M MACHINE"**
- Press **[T,S,M]** softkey.
- Place the cursor on the blank **"T".**
- Type the number of tool to be loaded. Ex: **01**
- Press **"INPUT"**
- Press **"CYCLE START"**

2º - Manually, unload the tool (of spindle):

3º - Access the tool list page.

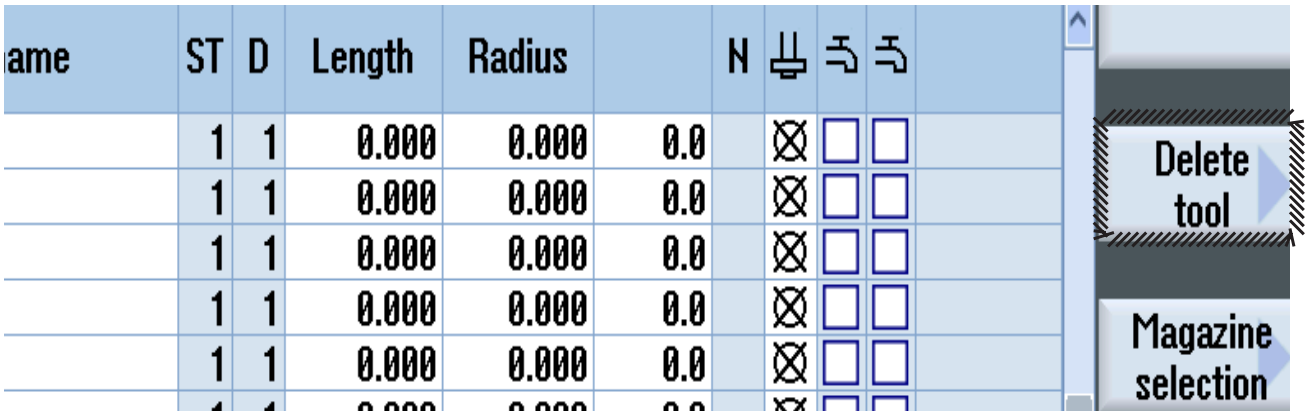
- Press **"OFFSET"**
- Press **[*Tool List*]** softkey.

4º - Uload the tool (of magazine):

- Place the cursor in the tool to erase.
- Press **[*UNLOAD*]** softkey.

5º - Delete the tool:

- Place the cursor in the tool to delete
- Press **[*DELETE TOOL*]** softkey.
- Press **[*OK*]**.



7.3 - TOOL PRESET

The preset of tools is a operation on which we identify the tool length, since the spindle face, until its tip. We can identify this length by using specific external measuring devices, or by operations done in the own machine.

7.3.1 – Tools present done in the machine

In order to determine the tool length in the own machine, we shall:

1st – By function ***"T,S,M"****, load tool "T00"*

- Press key **"JOG"**.
- Press key **"M MACHINE"**.
- Press softkey **[T,S,M]**.
- Place the cursor in field "T" and type "0".
- Press key **"INPUT"**.
- Press key **"CYCLE START"**.

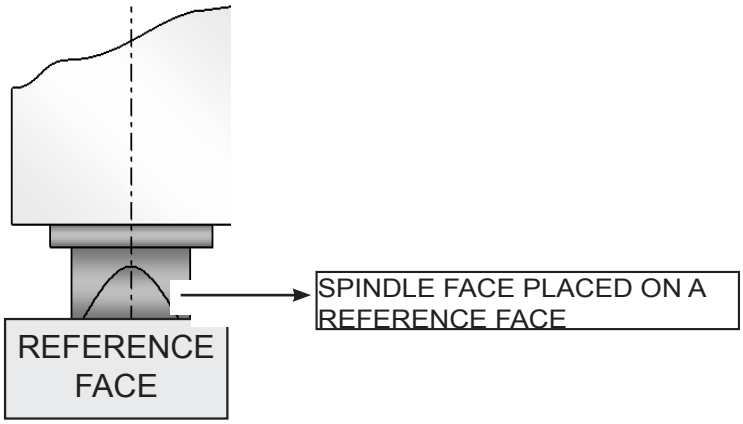
2nd – Erase the values contained in point zero G54

- Press key **"OFFSET"**.
- Press softkey **[DESL. ZERO]**
- Press softkey **[G54 ~ G599]**
- Place the cursor on axis Z of option G54.
- Type value "0".
- Press key **"INPUT"**.

3rd – By the electronic handle, touch the face of the spindle on a reference surface (part face, table face or device face):

- Press **"MMACHINE"**.
- Press key **"JOG"**.
- On the remote panel, select the axis you want: **"X, "Y", "Z"**
- On the operation panel, select the infeed you want by the keys: **[1], [10], [100], [1000]**.
- Keep pressed the buttons located in the side part of the remote panel.

Perform axis displacement by the handle, observing the direction (**+/-**), up to you touch the tool on the reference surface.



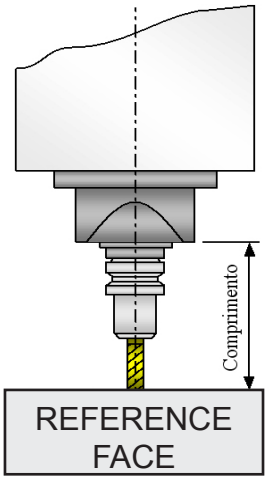
NOTES**: To desable the remote panel is necessary select the increase [0] and press **"ENABLE REMOTE PANEL".

4th – define a zero point on the reference surface:

- Press "**MACHINE**".
- Press softkey ***[MEDIR PEÇA]***.
- Press softkey ***[BORDA]***.
- Press softkey **[Z]**
- Place the cursor on field "**desl. Zero"** and press key **"SELECT"**, in order to select the zeroing you want (G54).
- Place the cursor on field "**direção**", and press key **"SELECT"**, until you select the correct direction (plus signal).
- Place the cursor on field "**Z0"** and type the value **"0"**.
- Press key **"INPUT"**.
- Press softkey **[DEF. NPV]**

5th – define the tool length

- Press key "**MACHINE"**.
- By the mode "**T,S,M**", load the tool you want to reference.
- By the electronic handle, touch the tool on the face where the spindle was referenced.
- Press softkey **[MEDIR FERR.]**
- Press softkey **[COMPRI. MANUAL]**
- On field **"PTO REF.**", press key **"SELECT"**, in order to choose option: **"PEÇA"**.
- On field **"Z0"**, type value "**0"** and press key **"INPUT"**.
- Press softke

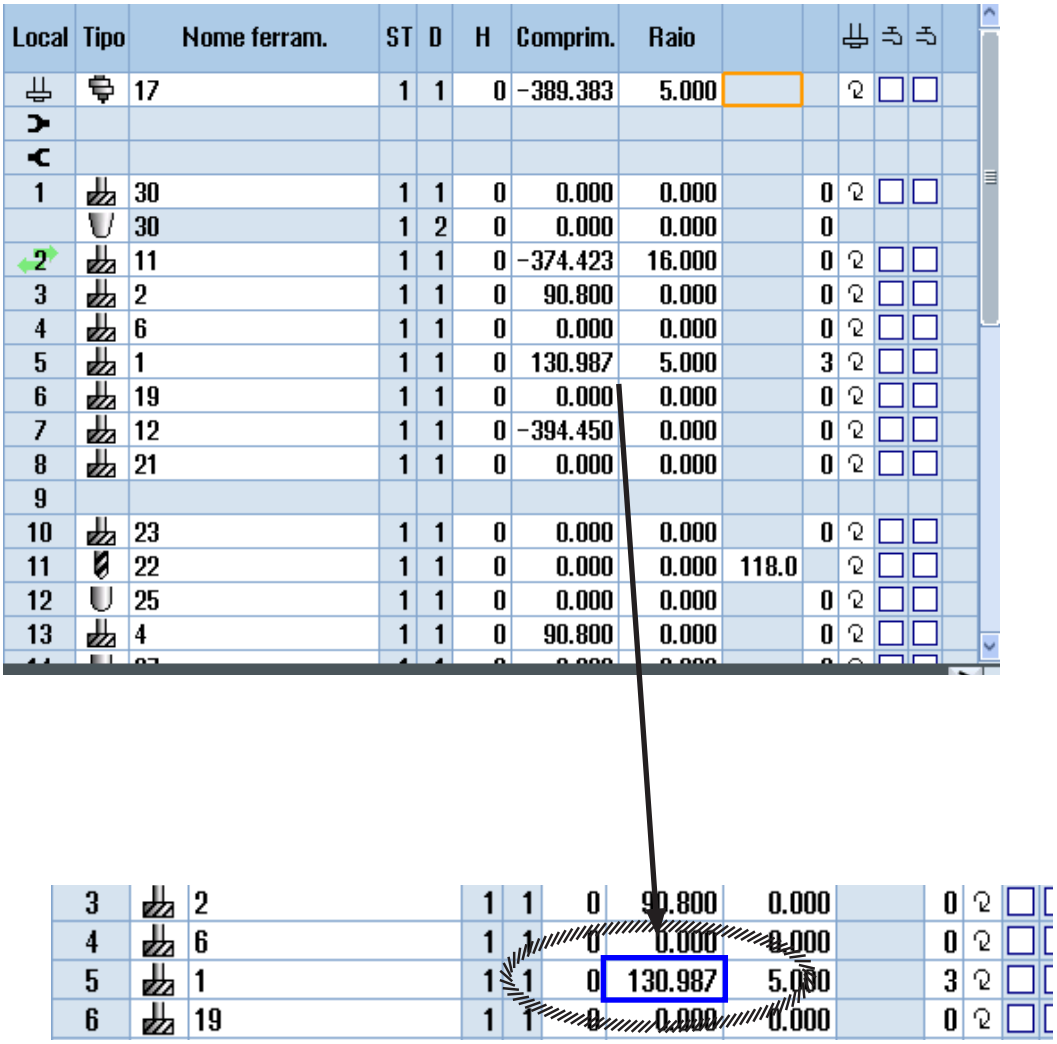


***OBSERVATION**: Using the procedure abovementioned, there is no need to make zero-part on axis "**Z"**. Thus, it is necessary to erase data contained on field **"Z"** of the page "**Desl. Zero**"(turn off zero)*
*When working with ball tip tool on **"XYZ"** plans (G18) or **"YZ"** (G19), in order to perform the radius offset in suitable mode, the reference of the tool shall be done by radius center.*
*Example: For a tool of "**Ø10mm**", we shall consider that the tool shall come downward **"5mm"** (radius value) more than the point touched on the part.*
*So, if the length registered, when the tip of the tool touches the part surface is "**-350.00mm**", the new value to be registered in the page "**Lista fer.**" (List of tools) Shall be "**355.00mm**".*
*When working on plans **"XZ"** or **"YZ"**, as well, the zeroing value of the tool shall not be registered in the field "**Compr. 1**" (Length 1), otherwise in the field "**Compr. 2**" (Length 2) (plan "**YZ**") or "**Compr. 3**" (plan "**XZ"**).*

7.3.2 - Tool referencing made out of the machine

This process is applied when the tool measuring is made in an external device. By doing this, the tool referencing is made only by loading its length value in the tool compensation page.

- Press **"OFF SET"**
- Press **[*Tool List*]** softkey.
- Place the cursor at the tool desired.
- Place the cursor at the column "length" and type a length value Ex.: **130.987**
- Press **"INPUT"**



NOTES: *The length values shall be entered unsigned.*

After entering all the tool lengths perform "workpiece reference" on "Z" axis.

The procedure above is applied to reference tools to work with radius compensation on XY (G17) plan. In case of working with spherical-tip tools with radius compensation on XZ (G18) and YZ (G19) planes, the tool referencing must be performed in the center of the sphere radius. To do that, subtract the tool radius value from its length value and enter this value as the referencing one.

Example: Length = 110.000
Tool radius = -5.000
Value to be entered = 105.000

After load the tool length it's necessary to do the workpiece reference in "Z".

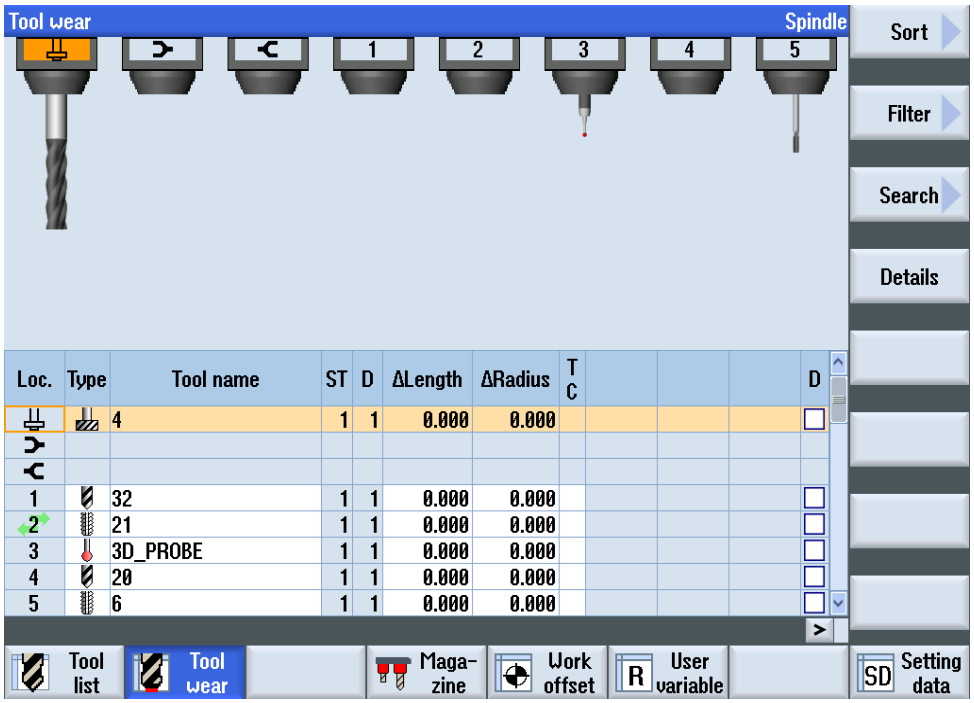
7.4 - FILL THE TOOLS DATA.

After do the tool reference in "Z" axis, it's necessary fill the tools data at the page "Tool List". To access this page it's necessary:

- Press **"OFFSET"**
- Press **[LIST]** softkey
- Place the cursor at the tool desired.
- Place the cursor at the column **"GROUPE"**.
- Use **"SELECT"** to select the groupe of tool.
- Place the cursor at the column "Tool name" will be the tool number, and this blank doesn't have to be changed.
- Place the cursor at the column ***"Ø"***.
- Type the tool diameter. Ex.: **5**
- Press **"INPUT"**
- When the tool group is a "**DRILL**", it's necessary place the cursor at the column "radius" and type the DRILL angle. Ex.: **118**
- Press **"INPUT"**
- When the tool group is a"**MILLING**", it's necessary place the cursor at the column "Number"
- Type the number of tooth (cut edge).
- Press "**INPUT"**
- Place the cursor at the blank "**Spindle right/left/off**" and press "select" to determinate the direction of spindle.
- Place the cursor at the blank "coolant1" and use the "**SELECT**" key to enable or desable the coolant.
- Place the cursor at the blank "coolant2" and use the "**SELECT**" key to enable or desable the coolant2 (high pression - 7bar).

8 - TOOL WEAR OFFSET.

All tools suffer progressive wear due to their friction with the removed material. Therefore, when the tool is applied for calibration, it is required to correct its wear to maintain product quality level related to the dimensional characteristics..



8.1 - TOOL WEAR COMPENSATION - "Z" AXIS.

- Press **"OFFSET"**.
- Press **[*TOLL WEAR*]** softkey.
- Use cursor keys (▲▼) to place the cursor on the tool desired.
- Use cursor keys (◄ ►) to place the cursor on the "length column" and press "**=**" key.
- Press**"+"** or **"-"** (according to a compensation desired).
- Type the value. Ex.: **0.1**.
- Press **"INPUT"**.

8.2 - TOOL WEAR COMPENSATION - "RADIUS"

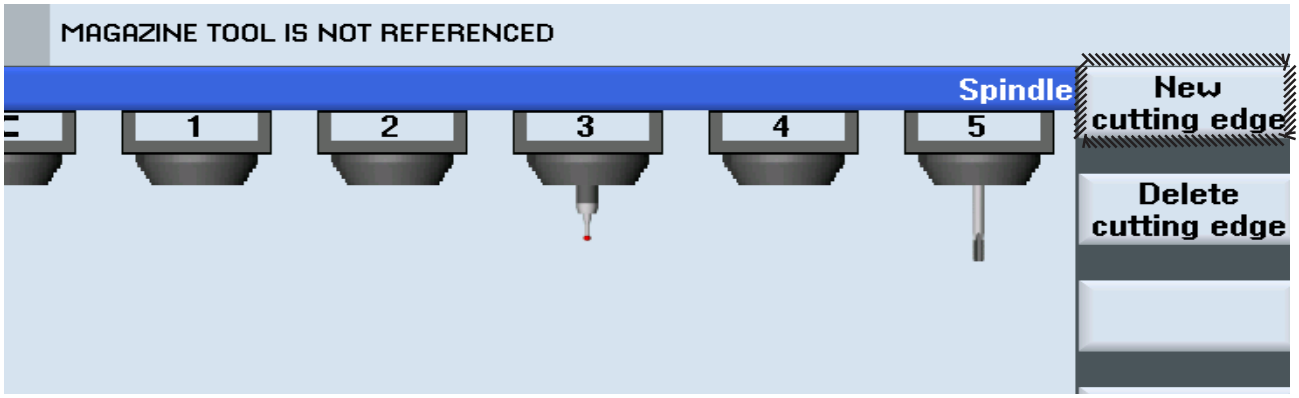
- Press **"OFFSET"**.
- Press **[*TOLL WEAR*]** softkey.
- Use cursor keys (▲▼) to place the cursor on the tool desired.
- Use cursor keys (◄ ►) to place the cursor on the "radius column" and press "**=**"
- Press **"+"** or **"-"** (according to a compensation desired).
- Type the value. Ex.: **0.1**.
- Press **"INPUT"**.

8.3 - PROCEDURE TO CREATE A NEW OFFSET.

To perform all tool arrangement and preset (process well kown as “SETUP”) it is required that tools and offset are already created.

The chapters below describe how to create and delete an offset.

- Press **“OFFSET”**
- Press **[*Tool List*]** softkey.
- Place the cursor at the tool desired and press the **[EDGES]** softkey.
- Press **[NEW CUTTING EDGE]**



To access this and others offsets it’s necessary:

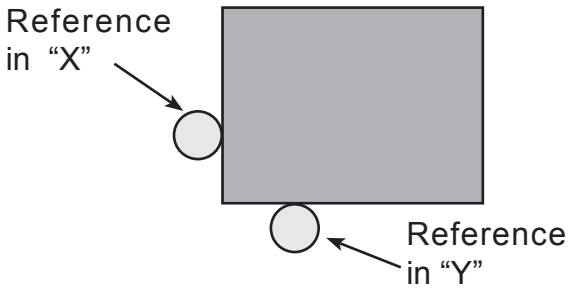
- Press **“OFFSET”**
- Press **[*Tool List*]** softkey.
- Place the cursor at the tool and offset desired.

9 - WORKPIECE REFERENCE DEFINITION

This procedure is applied to do a new reference point. This point it's called workpiece reference. In the "D" line - Siemens 828 it's possible to work with 99 offsets. They are: G54 ,G55,G56,G57,G58,G59,G507,G508...G599

9.1 -WORKPIECE REFERENCE ON PART EDGE.

This procedure it's applied to do the reference on part edge, according to the figure below:



To do this procedure it's necessary:

1º - Using the **"T,S,M"** *function, load the tool to be referenced:*

- Press **"JOG"**
- Press **"M MACHINE"**
- Press **[T,S,M]** key.
- Place the cursor at the blank "T" and type the tool number to be loaded Ex.: **1**
- Press **"INPUT"**
- If necessary, place the cursor at the blank "D" and type the offset number. Ex: 1
- Press **"INPUT"**
- Press **"CYCLE START"**

NOTES: *The tool applied to do a workpiece reference must has the radius value filled at the tool list offset PAGE.*

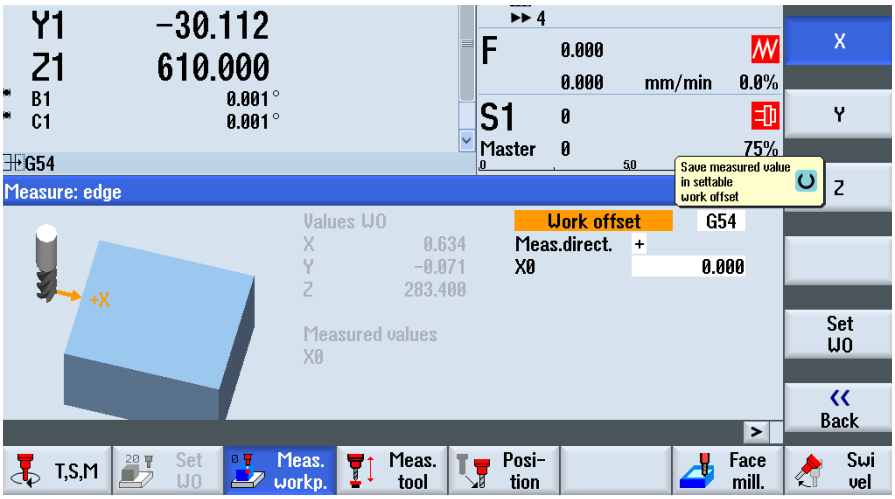
2º - Touch the tool on the part surface to be applied as reference:

- Press **"M MACHINE"**
- Press **"JOG"**
- Press **"ENABLE REMOTE PANEL"**
- In the remote panel, select the axis desired: **"X","Y", "Z"** or **"4"** (optional).
- In the remote panel, select the increase desired, using the **[1], [10], [100], [1000]**.
- Turn the crank while holding the safety button pressed (located on the remote

panel side) and touch the milling in the side od the part.

3º - Reference the workpiece.

– Press "**MACHINE**"
– Press **[*MEAS. WORKP.*]** softkey.
– Press softkey
– Select the axis desired using the "**X**" or "**Y**" softkey.
– Place the cursor in front of "Work Offset" and select the reference G54 or others desired using the "**SELECT**" key.
– Place the cursor at the blank "***meas. direct.***" and press the "**SELECT**" to choose the direction.
– Place the cursor at the blank **"X0"** or **"Y0"** according to axis desired and type the distance between the tool and the workpiece - Ex: 0
– Press "**INPUT**"
– Press ***[SET. WO]*** softkey



NOTES:
Case the point where the tool touch the part is not the same point then workpiece reference, it's necessary fill the offset value in the "X0" or "Y0" blank (according to the axis to be referenced.)

9.2 - WORKPIECE REFERENCE IN INCLINED PART.

– Press "**MACHINE**" key.
– Press **[*MEAS. WORKP.*]** softkey.
– Press softkey.
– Place the cursor in front of "WORK OFFSET" and press the "**SELECT**" to select the workpiece reference desired (G54 - G59 e G505 - G599).

– Place the cursor at the blank “X0” and type the distance between the workpiece and the reference - “**X**”. Ex: 0.

– Place the cursor at the blank “X0” and type the distance between the workpiece and the reference - “**Y**”. Ex: 0.

– Use the remote panel to touch the tool in the side of part as shown in figure by the side.

– Press **[SAVE P1]** softkey.

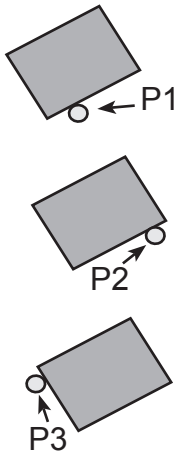
– Use the remote panel to touch the tool in the side of part as shown in figure by the side.

– Press **[SAVE P2]** softkey.

– Use the remote panel to touch the tool in the side of part as shown in figure by the side.

– Press **[SAVE P3]** softkey.

– Press **[*SET WO*]**



NOTES: *The tool for workpiece reference must has the radius value filled at the tool list offset PAGE.*

*To change the edge of part it’s necessary press the “**SELECT**” key and select POS 2”, “POS 3” or “POS 4” according to the edge desired.*

Case the point where the tool touch the part is not the same point that workpiece reference, it’s necessary fill the offset value in the “X0” or “Y0” blank (according to the axis to be referenced.)

The inclination angle of part will be introduced automatically at the blank “α 0” and will be saved in the workpiece reference PAGE.

9.3 - WORKPIECE REFERENCE IN A HOLES CENTER.

– Press “M MACHINE”

– Press **[*MEAS. WORKP.*]** softkey.

– Press softkey.

– Use the remote panel to touch the tool in the “Y” negative side of hole as shown in figure by the side.

– Press **[SAVE P1]** softkey.

– Use the remote panel to touch the tool in the “Y” positive side of hole as shown in figure by the side.

– Press **[SAVE P2]** softkey.

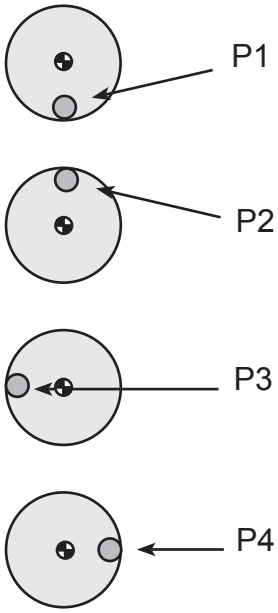
– Use the remote panel to touch the tool in the “X” negative side of hole as shown in figure by the side.

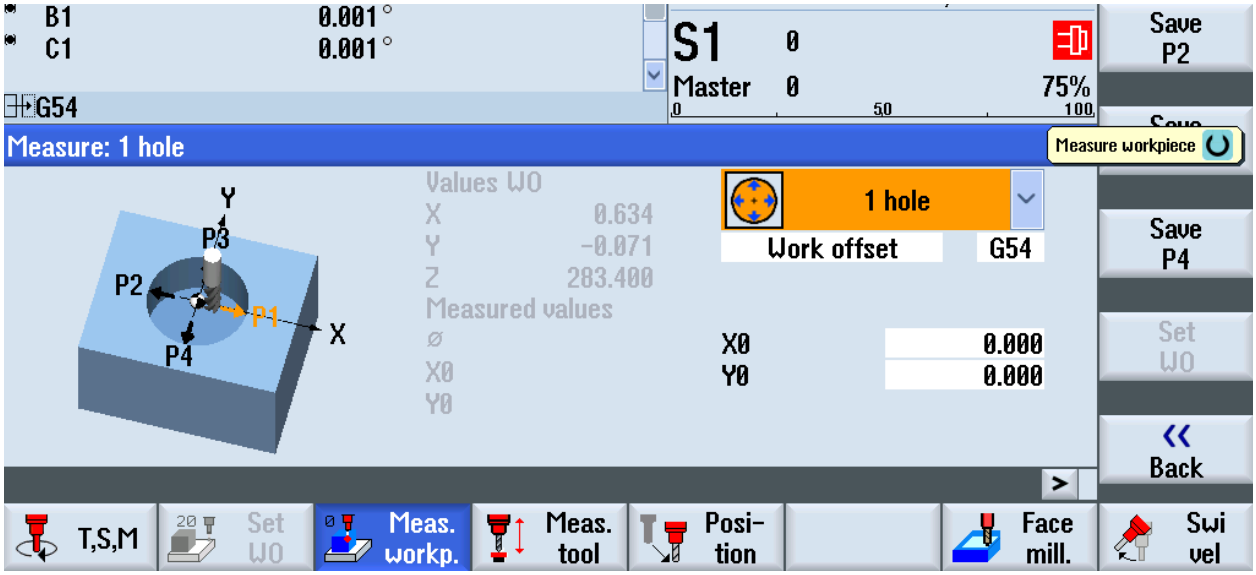
– Press **[SAVE P3]** softkey.

– Use the remote panel to touch the tool in the “X” positive side of hole as shown in figure by the side.

– Press **[SAVE P4]** softkey.

– Press **[SET WO]** softkey.





NOTES:*The tool applied to do a workpiece reference must has the radius value filled at the tool list offset PAGE.*

The part diameter value will be introduced automatically at the blank ø.

Case necessary offset the reference point fill the value in the “X0” or “Y0” blank (according to the axis to be referenced.)

The reference will be introduced automatically and will be saved in the workpiece reference PAGE.

9.4 - WORKPIECE REFERENCE IN A CIRCULAR SPIGOT.

– Press **[MACHINE]** softkey.

– Press **[MEAS. WORKP.]** softkey.

– Press softkey.

– Use the remote panel to touch the tool in the “X” negative side of pin as shown in figure by the side.

– Press **[SAVE P1]** softkey.

– Use the remote panel to touch the tool in the “X” positive side of pin as shown in figure by the side.

– Press **[SAVE P2]** softkey.

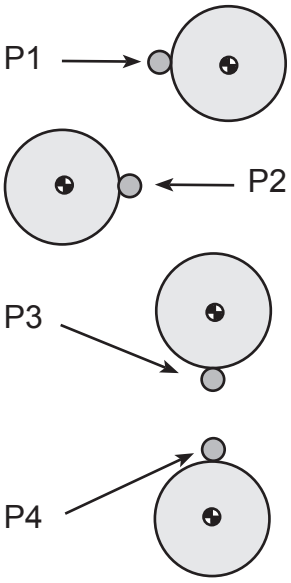
– Use the remote panel to touch the tool in the “Y” negative side of pin as shown in figure by the side.

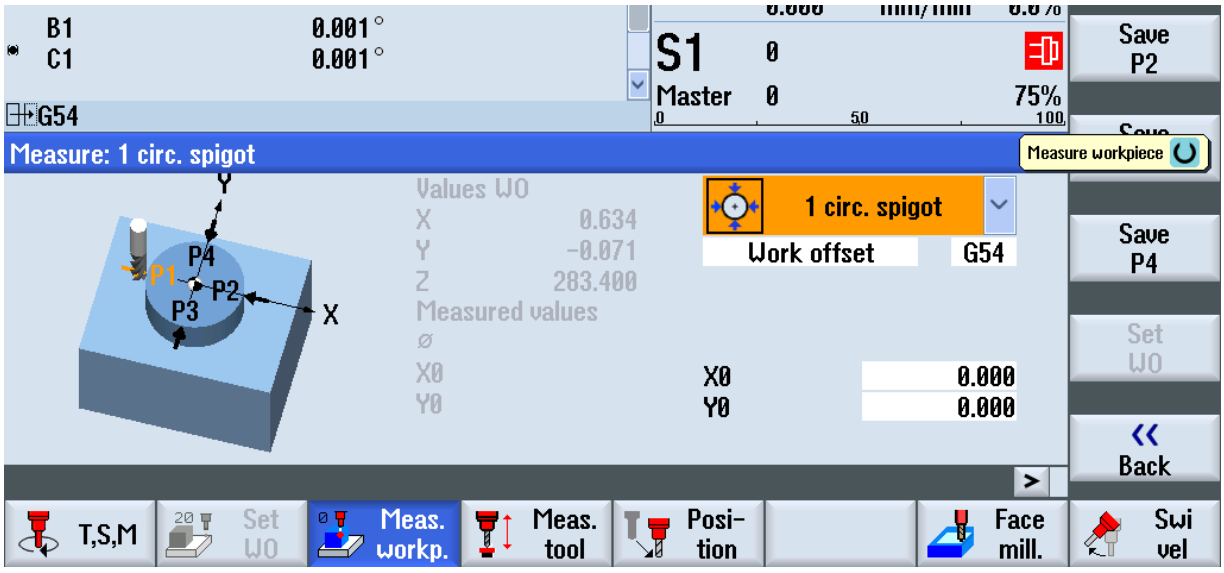
– Press **[SAVE P3]** softkey.

– Use the remote panel to touch the tool in the “Y” positive side of pin as shown in figure by the side.

– Press **[SAVE P4]** softkey.

– Press **[SET WO]** softkey.





NOTES:*The tool applied to do a workpiece reference must has the radius value filled at the tool list offset PAGE.*

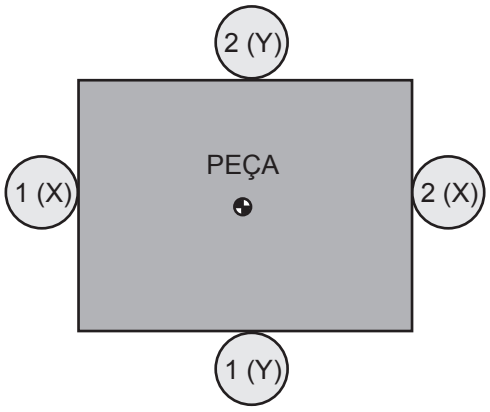
The diameter value will be introduced automatically at the blank ø.

Case necessary offset the reference point fill the value in the “X0” or “Y0” blank (according to the axis to be referenced.)

The reference will be introduced automatically and will be saved in the workpiece reference PAGE.

9.5 - WORKPIECE REFERENCE IN CENTER OF PART.

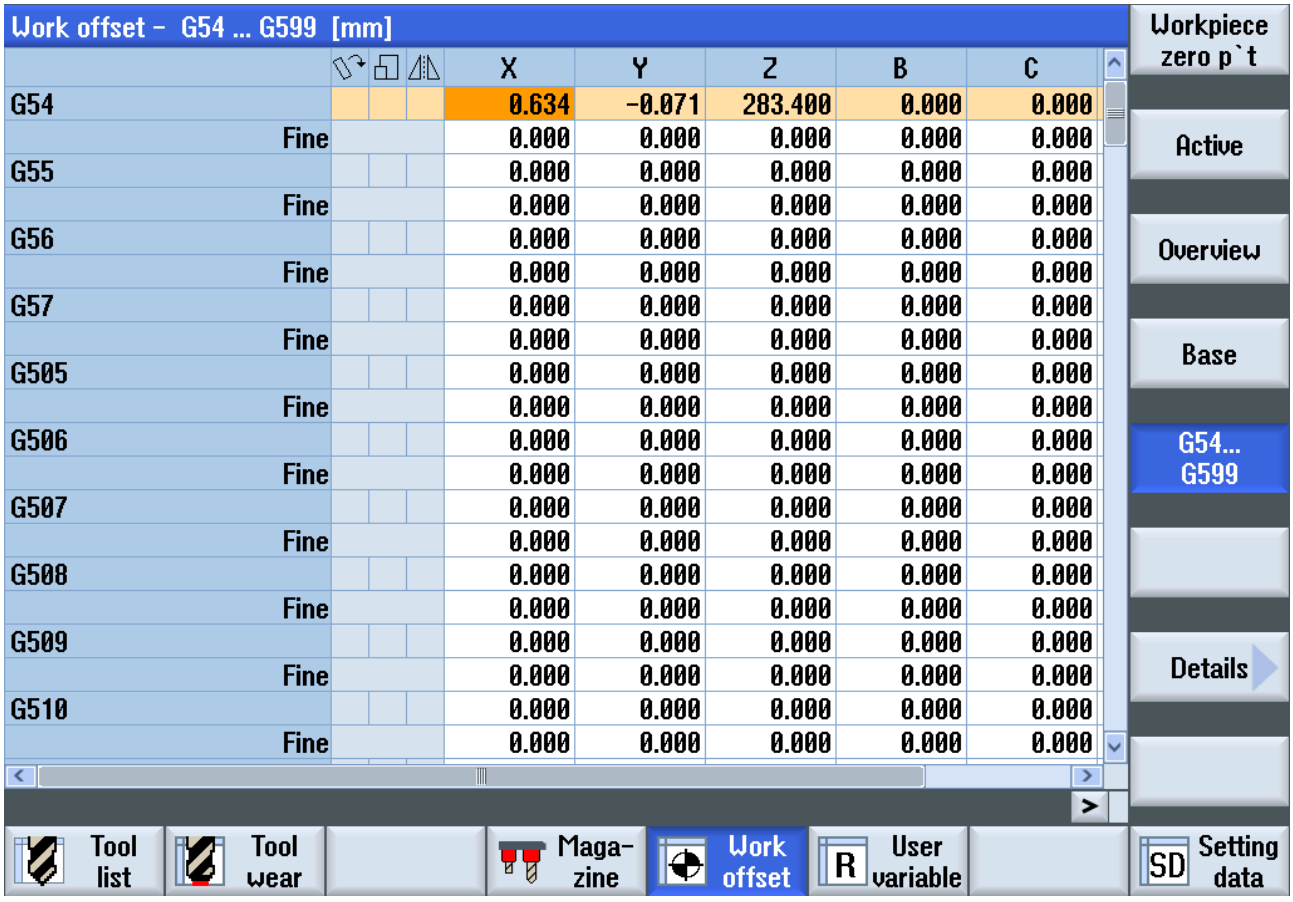
– Press **“JOG”**
– Press **“WCS/MCS”**
– Press **[M MACHINE]**
– Press **[T,S,M]** softkey.
– Place the cursor at the blank “WORK OFFSET” and press the “**SELECT**” key to choose the reference desired.
– Press **“CYCLE START”** key.
– Place the tool in the side of part.
– Press **[*SET WO*]** softkey.
– Press the **[X=0]** or **[Y=0]** softkey according to a axis desired.
– Place the tool in a opposite side.
– Press **[*SET WO*]** softkey.
– Use the arrows (↑ ↓ → ←) to place the cursor in a axis to do the reference. (**“X”** or **“Y”**).
– Press the “ **=** ” softkey.
– Type “ ***/2*** ” (divide by 2)
– Press **“INPUT”**



NOTES: *The reference will be introduced automatically and will be saved in the workpiece reference PAGE.*

9.6 - WORKPIECE REFERENCE (G54 - G59 E G505 - G599) ADJUST

– Press **[OFFSET]** softkey.
– Press **[WORK OFFSET]** softkey.
– Place the cursor at the line according to a reference desired (“**X**”, “**Y**”, ou “**Z**”).
– Press the “ **=** ” key.
– Press “+” or “-” according to direction of compensation.
– Type the compensation value. Example: **0.2**
– Type **“INPUT”**



10 - PROGRAM EXECUTION

10.1 - RUN A PROGRAM FROM MACHINE MEMORY

- Press ***"PROGRAM MANAGER".***
- Press ***[NC]*** softkey .
- Select the program., using the cursor ►, ◄, ▲ or ▼.
- Press ***"INPUT".***
- Press **[*EXECUTE*]** softkey.
- Press **[*CONT. PROG.*]** softkey.
- "**RELEASE**" the oprtions [***PROGRAM TEST***] and [***DRY RUN***].
- Press **[*BACK*]** softkey.
- Press **"*CYCLE START*"**.

10.2 - RUN A PROGRAM FROM USB (MODE 1)

- Press ***"PROGRAM MANAGER".***
- Press ***[USB]*** softkey.
- Select the program., using the cursor ►, ◄, ▲ or ▼.
- Press ***"INPUT".***
- Press **[*CALL*]** softkey.
- Press "**CYCLE START"**

10.3 - EXECUTING A PROGRAM FROM USB (MODE 2).

The program can be executed directly from memory card doing a call in the program located in the machine memory. To do this it's necessary:

- Create a new program in the machine memory.
- Insert in a program the syntax: **EXTCALL("USB:\File Name .File Extension")**.
- Run the program.

Example: If a program in the card has the name EXERCISE.MPF, it's necessary insert:

EXTCALL("USB:\EXERCISE.MPF")

10.4 - ABORT PROGRAM EXECUTION

– Press "**CYCLE STOP**"
– Press "**RESET**"

10.5 - RESTARTING IN THE MIDDLE OF PROGRAM.

10.5.1 - Restarting in the middle of program (Shopmill).

– Press **"AUTO"**
– Press **[*MACHINE*]**.
– Use cursor keys (◄, ►, ▲, ▼), to place the cursor on the tool desired.
– Select the program, using the cursor ►, ◄, ▲ or ▼.
– Press **[BLOCK *SEARCH*]** softkey.
– Press **[*START SEARCH*]** softkey.
– (CNC will show the mesage **"*SEARCH TARGET REACHED. COLLECTED FUNCTIONS WILL BE EXECUTED AFTER CYCLE START.*"**).
– Press **"CYCLE START"**
– Press **"CYCLE START"** again.

10.5.2 -Restarting in the middle of program (ISO Mode).

– Select the program
– Press **"AUTO"**
– Press **[MACHINE]**
– Place the cursor in the cycle desired.
– Press **[BLOCK *SEARCH*].**
– Press ***[BLOCK SEARCH MODE].***
– Select the Mode.
– Press [**START SEARCH**] softkey.
– (CNC will show the mesage **"*SEARCH TARGET REACHED. COLLECTED FUNCTIONS WILL BE EXECUTED AFTER CYCLE START.*"**).
– Press **"CYCLE START"**
– Press **"CYCLE START"** again.

NOTES: *To start in the middle of program in a cycle wtih many position (using ShopMill) it's necessary follow procedure below:*

10.5.3 - Start in the midle of program using Shopmill (Cycles with many position).

- Press **"AUTO"**
- Press **[M MACHINE]**.
- Place the cursor at the block - start position of cycle .
- Press ***[BLOCK SEARCH]*** softkey.
- Press **[*START SEARCH*]** softkey.
- Place the cursor at the column **"Number of start hole"** and type the number of initial hole desired. Ex:2
- Press **[*OK*]** softkey.
- (CNC will show the mesage ***"SEARCH TARGET REACHED. COLLECTED FUNCTIONS WILL BE EXECUTED AFTER CYCLE START."**).
- Press **"CYCLE START"**
- Press **"CYCLE START"** again.

10.5.4 - Start in the middle of program interrupted during the execution.

- Press **"AUTO"**
- Press **[M MACHINE]** softkey.
- Place the cursor at cycle desired.
- Press **[SEARCH BLOCK]** softkey.
- Press **[INTERRUPT POINT]** softkey.
- Press **[*START SEARCH*]** sofktey.
- (CNC will show the mesage ***"SEARCH TARGET REACHED. COLLECTED FUNCTIONS WILL BE EXECUTED AFTER CYCLE START."**).
- Press **"CYCLE START"**
- Press **"CYCLE START"** again.

10.5.5 - Start in the middle of program in the USB

- Press ***"PROGRAM MANAGER".***
- Press ***[USB]*** softkey.
- Use the keys ►, ◄, ▲ and ▼ **to** access the program to run.
- Press ***"INPUT".***
- Press ***[CALL]*** softkey.
- Press ***[BLOCK SEARCH]*** softkey,
- Press **[*SEARCH TEXT*]** softkey.
- Type the information to search in the column "TEXT". Ex: **T01**
- Press **"INPUT"**
- Press **[START SEARCH]** softkey.
- Press **"CYCLE START"** (CNC will show the mesage **"TO CONTINUE THE PROGRAM PRESS CYCLE START"** and will start the beep alarm).
- Press "**CYCLE START"** again.

10.6 -INTERRUPT / RESTART PROGRAM EXECUTION.

To interrupt the program execution, to change inserts, for cleaning or any purpose, follow the steps below:

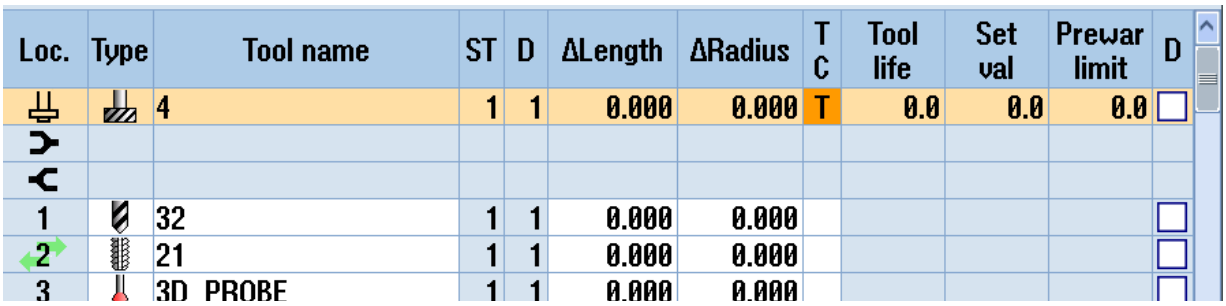
- During the execution press **"CYCLE STOP"**.
- Press **"JOG"**.
- Select the axis desired (**"X"**, **"Y"** or **"Z"**).
- Press "**+**" or "**-**" , according to direction desired.
- Stop the spindle using the **"SPINDLE STOP"** (if necessary).
- Do the necessary operation (clean the part, change inserts, etc...)
- Turn the SPINDLE using **"SPINDLE RIGHT"**.
- Press **"REPOS"**.
- Select the axis desired (**X**,**Y** ou **Z**) to back the interruption position.
- Press **"+"** or **"-"** to back in the initial coordinate
- Press **"AUTO"**.
- Press **"CYCLE START"**.

NOTES: *If the operator doesn't return the axis to interruption position, this will be made automatically by the "CYCLE START" key. During the return, the 3 axis will be moved simultâneously.*

11 - TOOL USEFUL LIFE MONITORING.

Tool useful life monitoring process establishes an execution limit (by time or by parts amount) for each tool and compels the machine to generate an alarm when this limit is reached. To limit the number of these executions, it is required to know how many parts (or how long) the tool is able to machine, before becoming unable for use (very high worn out).

11.1 - USEFUL LIFE MONITORING BY PARTS AMOUNT.



Loc.	Type	Tool name	ST	D	ΔLength	ΔRadius	T C	Tool life	Set val	Prewar limit	D
		4	1	1	0.000	0.000	T	0.0	0.0	0.0	
1		32	1	1	0.000	0.000					
2		21	1	1	0.000	0.000					
3		3D PROBE	1	1	0.000	0.000					

To operate with tool life monitoring by parts amount, it is required::

- Press **“OFFSET”**.
- Press **[*DESG. FER.*]** softkey .
- Use cursor keys (◄, ►, ▲, ▼) to place the cursor on the line of desired tool number.
- Use cursor keys (◄, ►) to place the cursor on **“T C ”**.
- Press **“SELECT”** to select the option “Q” (part quenatity).
- Use cursor keys (◄, ►, ▲, ▼) to place the cursor on “CMD VALUE”
- Type the quantity desired. Ex: **10**
- Press **“INPUT”**.
- Use cursor keys (◄►▲▼) to place the cursor on the “PREWAR LIM.” This blank shown the limit to prewarnig limit tool life. Example: **2**
- Press ***[ACTIVE]*** softkey
- Press **“INPUT”**.
- Repeat the same procedure to another tools.

After realize the procedure described above, it’s necessary insert in the program the syntax **“SETPIECE (X)”.** In this syntax, the letter “**X**” should be changed by the quantity of parts to works with the tool desired. Ex: ***SETPIECE (1)***

NOTES: *When the Pre-Warning limit tool life is over, the message “Tool X Number 0001 Pre-Warning limit tool life is over ” will shown.*

When the tool life is over, the messagem “Tool X Number the tool life limit is over ” will shown. If the program will be executed after the limit over, a new message is shown: “Tool life Manager: “No more tools prepared to work”

11.2 - LIFETIME MONITORING IN MINUTES.

To work with tool lifetime monitoring in minutes, proceed as follows:

- Press **“OFFSET”**.
- Press **[*TOOL WEAR*]** softkey.
- Use cursor keys (◄, ►, ▲, ▼) to place the cursor on the line of desired tool number**.**
- Use cursor keys (◄, ►) to place the cursor on **“$\frac{T}{C}$”**.
- Press **“SELECT”** to select the option “T” (tool life minutes).
- Use cursor keys (◄, ►, ▲, ▼) to place the cursor on “TOOL LIFE”, and enter time (in minutes) to keep the tool machining parts in machining motion.
- Use cursor keys (◄►▲▼) to place the cursor on the “PREWAR LIM.” This blank shown the limit to prewarnig limit tool life. Example: **2**
- Press ***[ACTIVE]*** softkey
- Press **“INPUT”**.
- Repeat the same procedure to another tools.

NOTE: *The monitoring time count in minutes occurs only when the machine performs a machining motion.*

When the Pre-Warning limit tool life is over, the message “Tool X Number 0001 over Pre-Warning limit tool life” will shown.

When the tool life is over, the messagem “Tool X Number over the tool life limit” will shown. If the program will be executed after the limit over, a new message is shown: “Tool life Manager: “No more tools prepared to work”